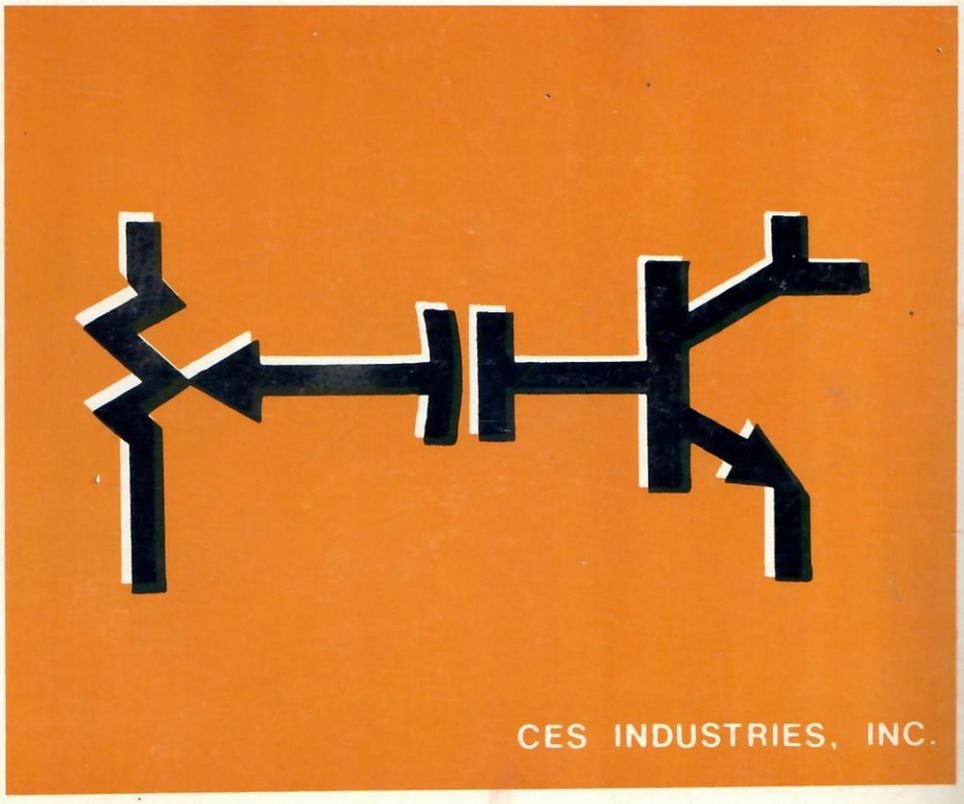
# Ed-Lab

## *EXPERIMENT MANUAL*

ELECTRICITY - ELECTRONICS BOOK II

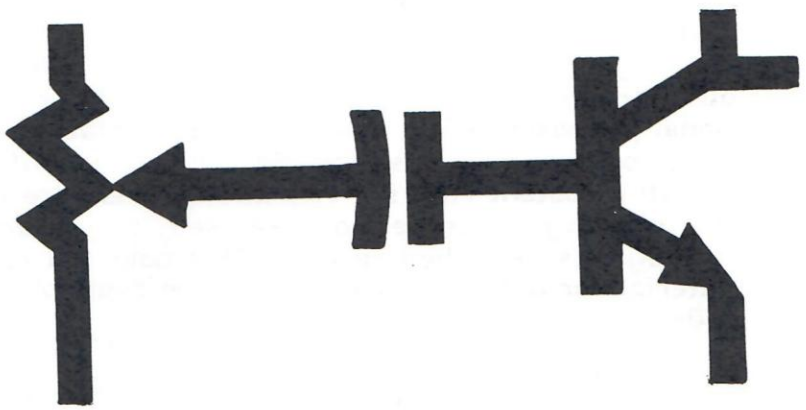


CES INDUSTRIES, INC.

# Ed-Lab

## *EXPERIMENT MANUAL*

**ELECTRICITY - ELECTRONICS BOOK II**



CES INDUSTRIES, INC.

Name _____

Class _____

School _____

# Ed-Lab

EXPERIMENT MANUAL

ELECTRICITY-ELECTRONICS BOOK II

POWER SUPPLIES, INDUSTRIAL CONTROLS,

TRANSISTORS, VACUUM TUBES;

UNITS 4,5,6,7

Copyright © 1974, 1976, 1980, 1981 CES INDUSTRIES, INC.

All rights reserved. No part of this book may be duplicated or reproduced in any form without the specific written permission of CES Industries, Inc. The use of this manual and the experiments described is restricted to performance with CES Ed-Lab training equipment and is prohibited from being used with any other training aid or electronic components or for other educational purposes. CES makes no representation that the circuits, logic, or programs described herein will not infringe on existing or future patent rights. There is furthermore no implication in the granting of licenses to make, sell, or use any circuits, logic, or programs described herein. This does not restrict use of any materials or data sheets which may be available from other sources.

CES INDUSTRIES, INC.
Educational Products Division
130 Central Avenue
Farmingdale, New York 11735
(516) 293-1420

ISBN 0-86711-014-7
Printed in United States of America

TABLE OF CONTENTS


1. NPN Transistor
2. Common Emitter D.C. Amplifier
3. Transistor and Beta Tester
4. Common Emitter Amplifier Biasing
5. A.C. Common Emitter Amplifier
6. C.E. Input and Output Impedance
7. Common Collector Amplifier or Emitter Follower
8. E.F. Input and Output Impedance
9. Emitter Bias and Degeneration
10. Emitter By-Pass Capacitor
11. Dynamic Amplifier and Phase Splitter
12. Transistor Characteristic Curve Tracer
13. Common Base Amplifier
14. Common Base A.C. Amplifier
15. Cascaded Amplifier
16. PNP Transistor
17. NPN - PNP Cascaded Amplifier
18. Constant Current Source
19. Difference Circuit
20. Difference Amplifier
21. Constant Current Difference Amplifier
22. Darlington Amplifier
23. Audio Driver
24. Feedback Amplifier
25. FET Operation
26. FET Common Source Amplifier
27. FET Common Drain Amplifier
28. FET Common Gate Amplifier
29. FET Characteristic
30. FET Constant Current Source

Ed-Lab

UNIT NO. 6

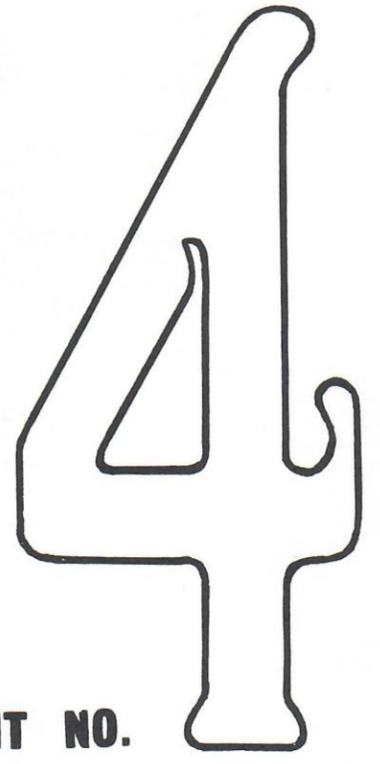
## TABLE OF CONTENTS

1. Triode D.C. Amplifier
2. Triode Characteristics
3. Plate Characteristic Curve
4. A.C. Triode Grid Bias Amplifier
5. Cathode Bias
6. Cathode Follower Amplifier
7. Diode Rectifier

# Ed-Lab

## *EXPERIMENT MANUAL*

**RECTIFIERS AND POWER SUPPLIES**



**UNIT NO.**

**CES INDUSTRIES, INC.**

| Ed-Lab | UNIT NO. 4 |
|---|---|

# EXPERIMENT MANUAL

# RECTIFIERS & POWER SUPPLIES

Copyright © 1974, 1976, 1980, 1981 CES INDUSTRIES, INC.

Name _____

Course _____

TABLE OF CONTENTS


| | |
|---|---|
| 1 | Transformer & Half-Wave Rectifier |
| 2 | Full-Wave Rectifier |
| 3 | Filter Capacitor |
| 4 | Bridge Rectifier |
| 5 | Regulation-Full wave |
| 6 | R.C. PI Filter |
| 7 | Full-Wave Voltage Doubler |
| 8 | Half-Wave Voltage Doubler |
| 9 | Voltage Tripler |
| 10 | Voltage Quadrupler |
| 11 | Zener Diode |
| 12 | Zener Characteristic Curve |
| 13 | Zener Regulator |
| 14 | Series Variable Voltage |
| 15 | Series Regulator |
| 16 | Regulation Measurement |
| 17 | Shunt Regulator |
| 18 | Feedback Regulation |
| 19 | Current Regulator |
| 20 | Op-Amp Regulator |
| 21 | Power Supply Efficiency |
| 22 | Collector Output Power |
| 23 | Darlington Series Regulator |

| Ed-Lab | TRANSFORMER & HALF-WAVE RECTIFIER | EXP. NO. 1<br>Page No. 1 |
|---|---|---|

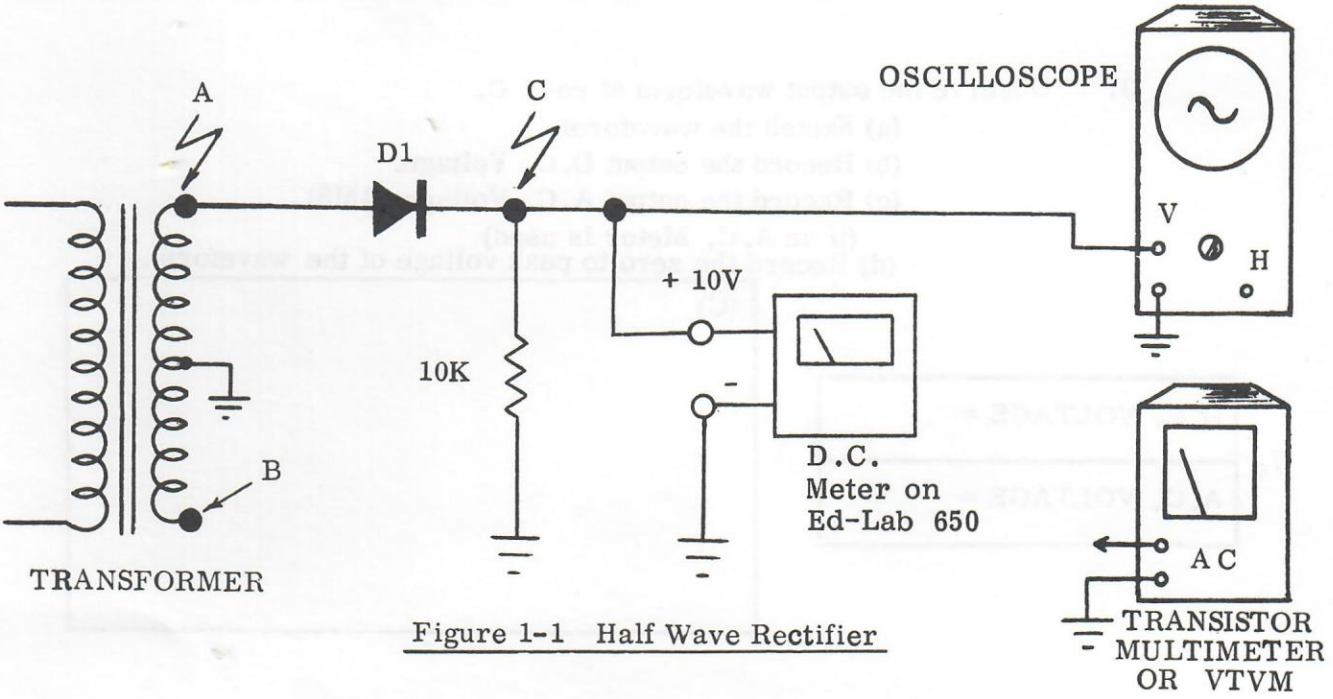



Figure 1-1 Half Wave Rectifier

Objective:

You will observe the output waveform from the TRANSFORMER (points A and B) and the output from the HALF-WAVE RECTIFIER DIODE (point C).

Procedure:

1. Wire the Half-Wave Rectifier circuit in Figure 1 - 1. Connect the Oscilloscope and the D. C. Meter on the Ed-Lab trainer. (You may optionally connect an external A.C. VTVM Multimeter.)

2. Observe the transformer waveforms at points A and B. Sketch the waveforms. Record the Period and the Peak-to-Peak Amplitude.

TRANSFORMER WAVEFORM (A)

(A)

TRANSFORMER WAVEFORM (B)

(B)

3. Observe the output waveform at point C.
   (a) Sketch the waveform.
   (b) Record the output D.C. Voltage.
   (c) Record the output A.C. Voltage (RMS) (if an A.C. Meter is used)
   (d) Record the zero to peak voltage of the waveform.

| $V_C$ | |
|---|---|
| | D.C. VOLTAGE = |
| | A.C. VOLTAGE = |

(C)

4. Reverse Diode D1
   (a) Sketch waveform
   (b) Record voltage (peak-to-peak)

| $V_C$ | |
|---|---|
| | D.C. VOLTAGE = |
| | A.C. VOLTAGE = |

| Ed-Lab | FULL WAVE RECTIFIER | EXP. NO. 2<br>Page No. 1 |
|---|---|---|

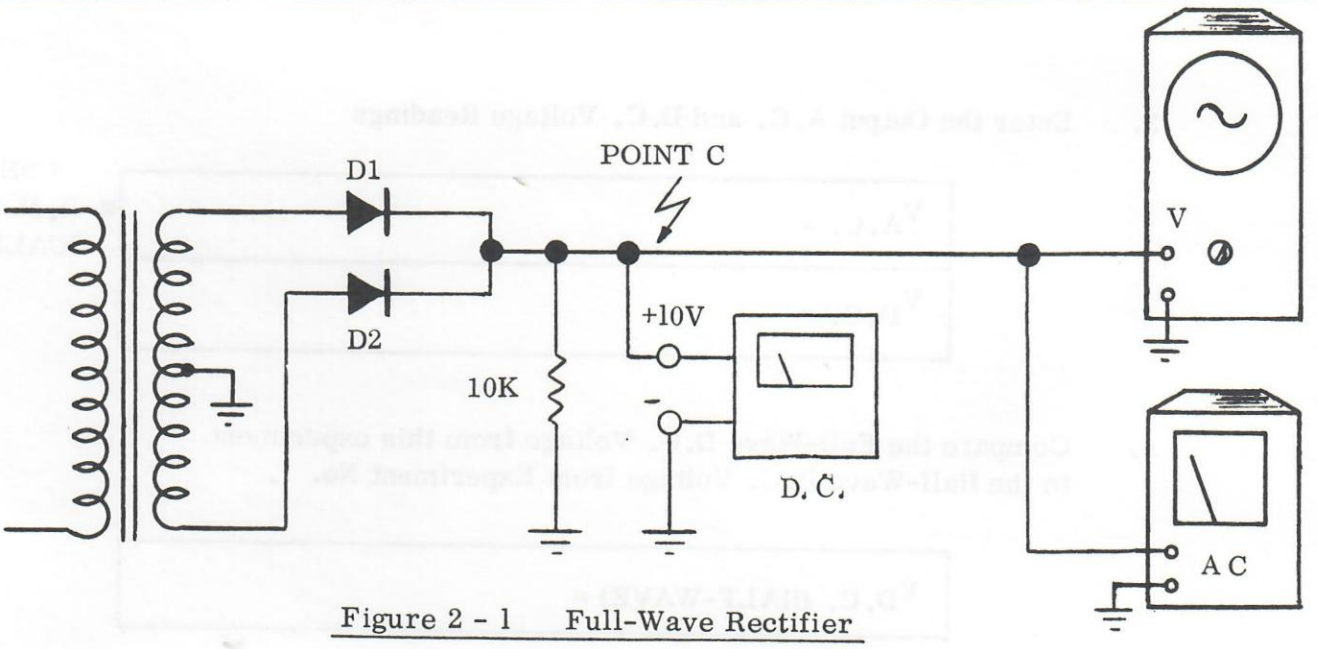



Figure 2 - 1 Full-Wave Rectifier

Objective:

You will use two diodes to provide both halves of the A. C. Sine Wave to provide a FULL-WAVE rectified voltage.

Procedure:

1. Wire the circuit in Figure 2 - 1.
2. Sketch the output waveform at Point C. Record the Peak-to-Peak Voltage.

OUTPUT
WAVEFORM (C)

3. Enter the Output A.C. and D.C. Voltage Readings

| | |
|---|---|
| $V_{A.C.}$ = | ← USE R.M.S. SCALE |
| $V_{D.C.}$ = | |

4. Compare the Full-Wave D.C. Voltage from this experiment to the Half-Wave D.C. Voltage from Experiment No. 1.

| |
|---|
| $V_{D.C.}$ (HALF-WAVE) = |

Enter Data from Experiment No. 1.

The Full-Wave D.C. Voltage output should be twice the Half-Wave D.C. Voltage output. Do your measurements agree?_____

**Ed-Lab** | FILTER CAPACITOR | **EXP. NO. 3** **Page No. 1**

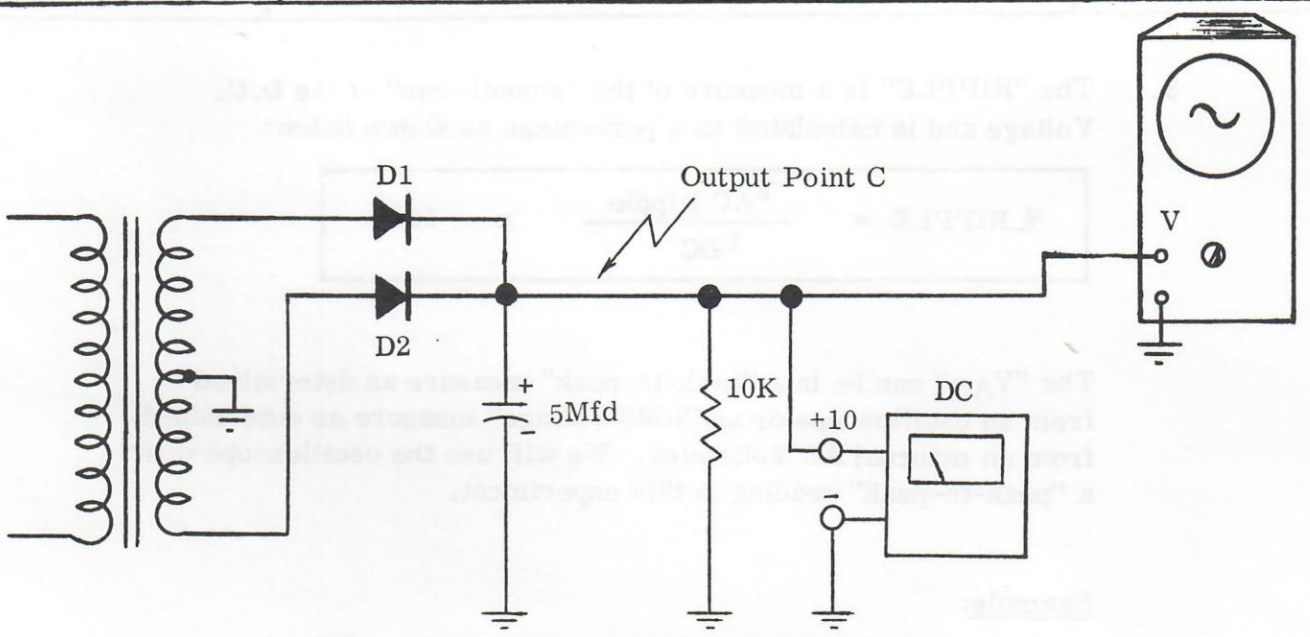



Figure 3-1 Rectifier with Filter Capacitor

Objective:

You will add a CAPACITOR to the output of the full-wave rectifier to provide a FILTERED D.C. Voltage. Various size capacitors will be used and you will measure the RIPPLE.

Procedure:

1. Wire the circuit in Figure 3 - 1 above.
2. (a) Sketch the waveform at Point C,
   (b) Measure the D.C. Voltage and
   (c) Measure the "peak-to-peak" A.C. Voltage on the Oscilloscope.

ENTER THE RESULTS IN THE TABLE BELOW FOR VARIOUS SIZE CAPACITORS.

| | METER | OSCILLOSCOPE | |
|---|---|---|---|
| CAPACITOR | $V_{D.C.}$ METER | $V_{A.C.}$ "RIPPLE" PEAK-TO-PEAK OSCILLOSCOPE | Sketch Ripple Waveform |
| 1 mfd | | | |
| 5 mfd | | | |
| 10 mfd | | | |
| 100 mfd | | | * |

*It will be necessary to set the SCOPE INPUT on A.C. and increase the vertical sensitivity to see this waveform.

3. The "RIPPLE" is a measure of the "smoothness" of the D.C. Voltage and is calculated as a percentage as shown below:

$$\% \text{ RIPPLE} = \frac{V_{AC} \text{ Ripple}}{V_{DC}} \times 100\%$$

The "$V_{AC}$" can be in a "peak-to-peak" measure as determined from an oscilloscope or an "RMS Voltage" measure as determined from an external AC Voltmeter. We will use the oscilloscope with a "peak-to-peak" reading in this experiment.

Example:

If $V_{AC}$ peak-to-peak = 0.2 volts

$V_{DC}$ = 8.4 volts

$$\% \text{ Ripple} = \frac{V_{AC}}{V_{DC}} \times 100\%$$

$$= \frac{0.2}{8.4} \times 100\% = \frac{20}{8.4}\%$$

$$= 2.4\%$$

Calcule the RIPPLE for each of the capacitor measurements. Enter the results below:

| CAPACITOR | PERCENTAGE RIPPLE "PEAK-TO-PEAK" |
|---|---|
| 1 mfd | |
| 5 mfd | |
| 10 mfd | |
| 100 mfd | |

What conclusion do you reach comparing the size of the Capacitor used and the resulting Ripple? What do you think will limit the size of the capacitor that you can use?

4. Summarize the Full-Wave Rectifier Voltages:

| | |
|---|---|
| Experiment No. 2 | $V_{DC}$ (no capacitance) = $V_{average}$ = |
| Experiment No. 3 | $V_{DC}$ (with 100 mfd) = $V_{peak}$ = |

5. Calculate the RATIO of $V_{PEAK}$ to $V_{AVERAGE}$:

$$\frac{V_{PEAK}}{V_{AVERAGE}} =$$

6. It can be determined theoretically that this ratio is 1.57. How does your calculation in paragraph 5 above compare to this theoretical number?_____

| Ed-Lab | BRIDGE RECTIFIER | EXP. NO. 4<br>Page No. 1 |
|---|---|---|

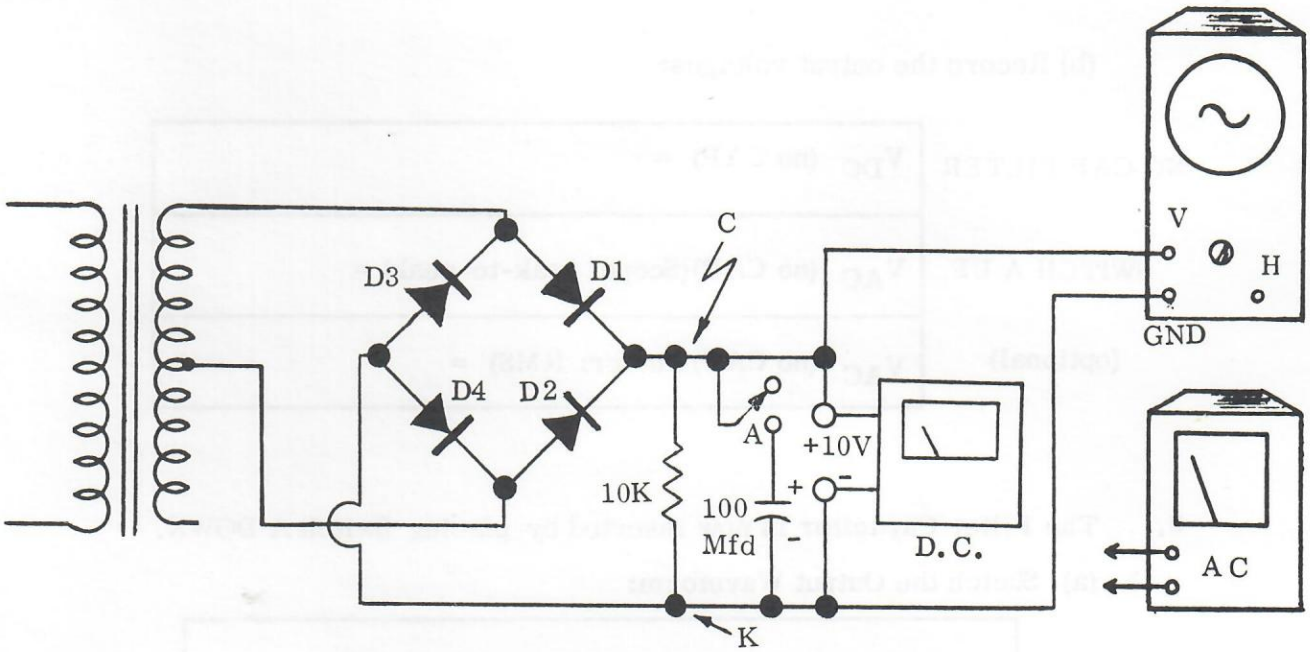



Figure 4 - 1 Bridge Rectifier

Objective:

You will use four diodes and a transformer "without using a center-tap" in a FULL-WAVE BRIDGE RECTIFIER circuit.

Procedure:

1. Wire the circuit in Figure 4 - 1.
   Use the oscilloscope to observe and measure the output waveforms. The Oscilloscope is used to measure the "peak-to-peak" ripple voltage and the D.C. Meter measures the power supply D.C. Voltage, and an external A.C. Meter (optional) measures the output voltage, RMS.
2. Observe the waveform on the oscilloscope, without the Filter Capacitor in the circuit; SWITCH A UP.
   (a) Sketch the observed output Waveform; between C and K. Record the Peak-to-Peak Voltage.

(b) Record the output voltages:

| | |
|---|---|
| NO CAP FILTER | $V_{DC}$ (no CAP) = |
| SWITCH A UP | $V_{AC}$ (no CAP)(Scope: peak-to-peak) = |
| (optional) | $V_{AC}$ (no CAP) (meter: RMS) = |

3. The Filter Capacitor is now inserted by placing Switch A DOWN.

(a) Sketch the Output Waveform:

(b) Record the output voltages

| | |
|---|---|
| WITH CAP | $V_{DC}$ (Filter cap) = |
| SWITCH A DOWN | $V_{AC}$ (Ripple)(scope: peak-to-peak) = |
| (optional) | $V_{AC}$ (Ripple)(meter: RMS) = |

4. How does the Output $V_{DC}$ (with Filter Cap) from (b) above compare between the FULL WAVE and BRIDGE Rectifiers?

| | | |
|---|---|---|
| BRIDGE | From (b) above | $V_{DC}$ (Filter Cap) = |
| FULL WAVE | From Exp. #3 | $V_{DC}$ (with 100 mfd Cap) = |

Are both voltages equal?

| Ed-Lab | REGULATION - FULL WAVE | EXP. NO. 5<br>Page No. 1 |
|---|---|---|

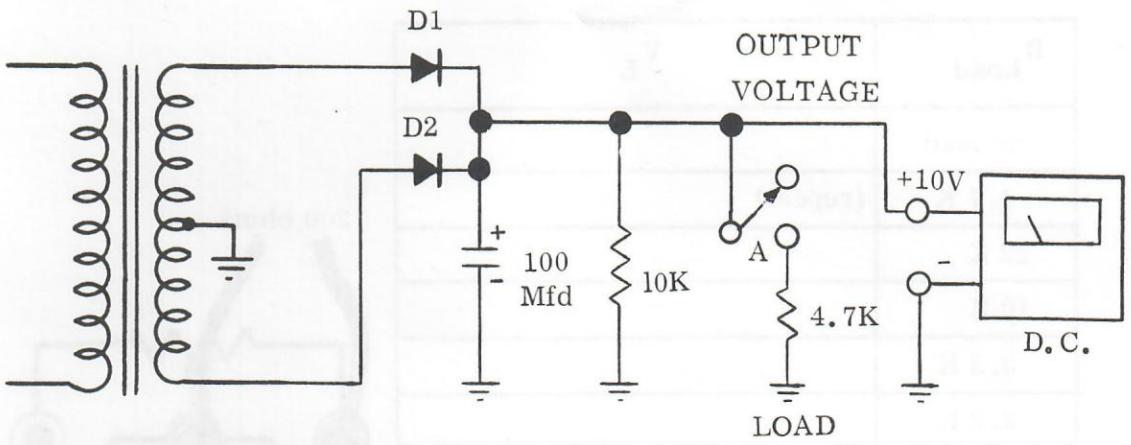



Figure 5-1 Full-Wave Rectifier with Load

Objective:

You will vary the load and measure the change in D.C. Voltage when a LOAD resistance is added to the "power supply" output. You will then calculate the REGULATION and OUTPUT RESISTANCE.

Procedure:

1. Wire the circuit.
2. Measure the NO-LOAD Voltage with switch A up:

| $V_{NL}$ = |
|---|

3. With the LOAD (switch A down), measure the output voltage.

| $V_L$ = |
|---|

4. Calculate the REGULATION.

$$\text{REGULATION} = \frac{(V_{NL} - V_L) \times 100\%}{V_L}$$

| REGULATION = |
|---|

5. Calculate the LOAD CURRENT.

$$I_L = \frac{V_L}{4.7\,K}$$

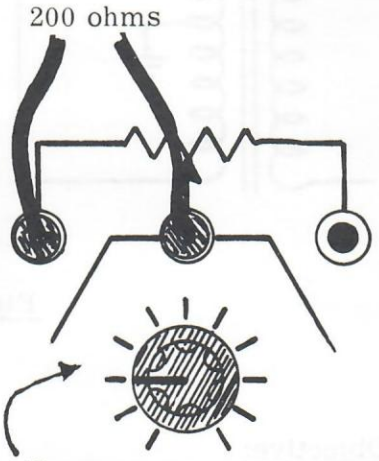
| $I_L$ = |
|---|

| Ed-Lab | REGULATION - FULL WAVE | EXP. NO. 5<br>Page No. 2 |
|---|---|---|

6. Measure the output voltage for different loads.

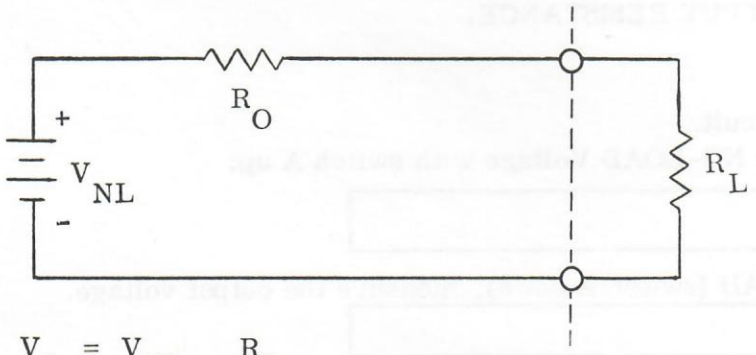
| $R_{Load}$ | $V_L$ |
|---|---|
| no load | |
| 4.7 K | (repeat) |
| 22 K | |
| 10 K | |
| 3.3 K | |
| 2.2 K | |
| 1 K | |
| 200* | |
| 100 | |

*Use 1K POT set at 2 divisions for 200 Ohms.




(Set Knob here for 200 ohms)

7. Calculate the OUTPUT RESISTANCE ($R_O$) using "Thevenin's Theorem" :




$$V_L = V_{NL} \frac{R_L}{(R_L + R_O)}$$

Then

$$R_O = R_L \frac{(V_{NL} - V_L)}{V_L}$$

Select R of 200 or 1K OHMs.
Calculate $R_O$

| $R_O$ = |
|---|

8. Calculate the REGULATION for the LOAD of 200 OHMS.

| REGULATION (200 ohm Load) = $\frac{(V_{NL} - V_L)}{V_L} \times 100\%$ = |
|---|

| Ed-Lab | R.C. PI FILTER | EXP. NO. 6<br>Page No. 1 |
|---|---|---|

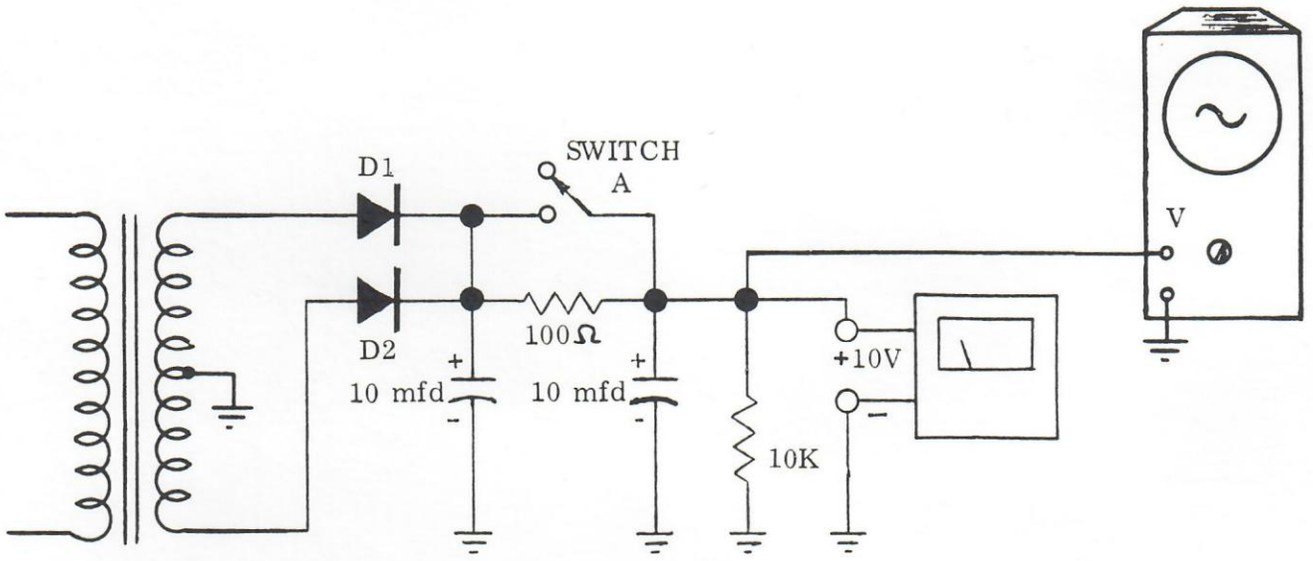



Figure 6-1 Rectifier with PI Filter

Objective:

You will use a PI FILTER with two capacitors and a resistor to reduce power supply ripple.

Procedure:

1. Wire the circuit. With Switch A down, the total filter capacitor size is 20 Mfd. With Switch A up, the total capacitor size is 20 Mfd consisting of two 10MFd capacitors separated by a 100 ohm resistor in a PI Filter.

2. Measure and observe the A.C. Ripple Voltage and enter the data below.

| | | measurement | | calculate |
|---|---|---|---|---|
| Switch A | Filter | A.C. pk to pk Ripple Voltage | D.C. Voltage | Percentage ripple |
| DOWN | CAPACITOR | | | |
| UP | PI (R-C) | | | |

3. Calculate the percentage ripple for both filters. Enter the result in the above table.

4. What is the advantage of the PI Filter?
Note that in many Power Supplies, an Inductor or CHOKE is used instead of the 100 ohm resistor.

| Ed-Lab | FULL-WAVE VOLTAGE DOUBLER | EXP. NO. 7<br>Page No. 1 |
|---|---|---|

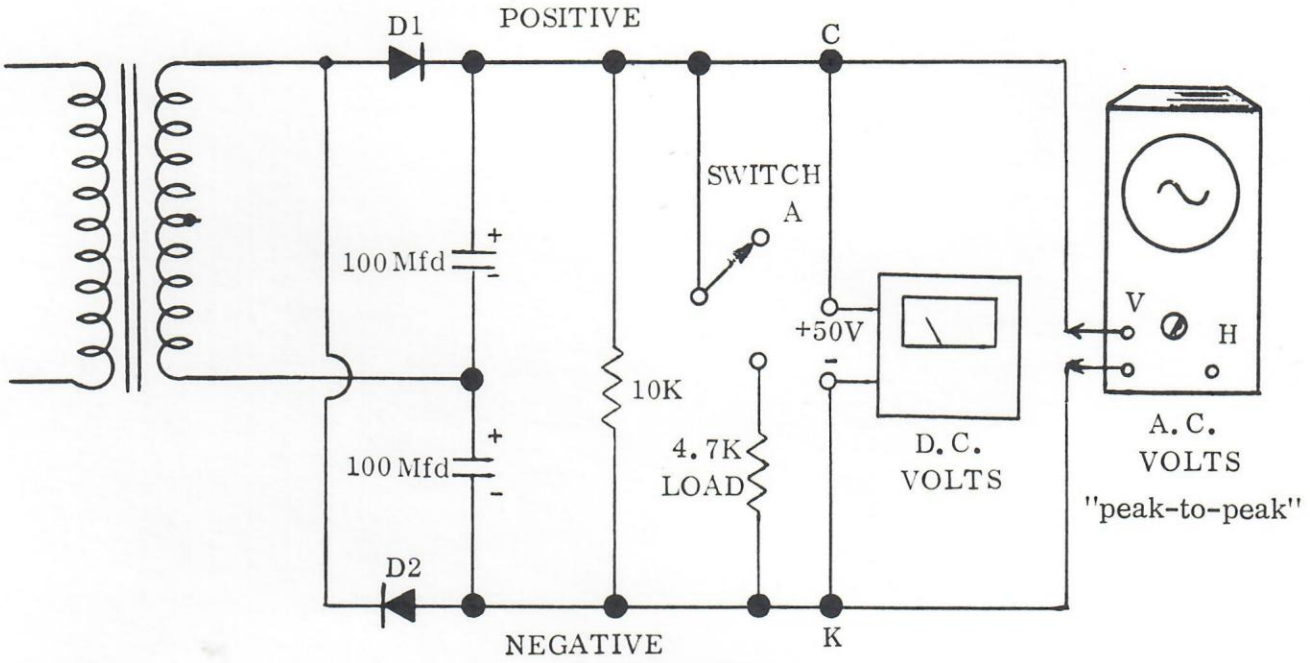



Figure 7-1 Voltage Doubler

Objective:

You will use a rectifier circuit which provides "double" the normal voltage output. You can consider the above circuit as two half-wave rectifiers for a positive and negative voltage. The output is thus twice the voltage. The above circuit will actually provide a theoretical voltage of 4 times the full-wave rectifier because the AC is across the entire transformer.

Procedure:

1. Wire the above circuit. The output is taken between points C and K.
2. With Switch A UP (NO-LOAD), read the output voltage and ripple.

| D.C. $V_{NL}$ = | A.C. $V_{NL}$ = |
|---|---|

3. With Switch A DOWN (4.7K LOAD), read the output voltage.

| D.C. $V_L$ = | A.C. $V_L$ = |
|---|---|

4. **Calculate the regulation :** $\dfrac{(V_{NL} - V_L)}{V_L}$ x 100% (D.C. Voltages)

| REGULATION = |
|---|

5. **Calculate the ripple under load:** $\dfrac{\text{A.C. } V_L}{\text{D.C. } V_L}$ x 100%

| RIPPLE = |
|---|

6. How does the Regulation for the Doubler compare with the Regulation for the Full Wave Rectifier (Experiment No. 5)?

| Ed-Lab | HALF-WAVE VOLTAGE DOUBLER | EXP. NO. 8<br>Page No. 1 |
|---|---|---|

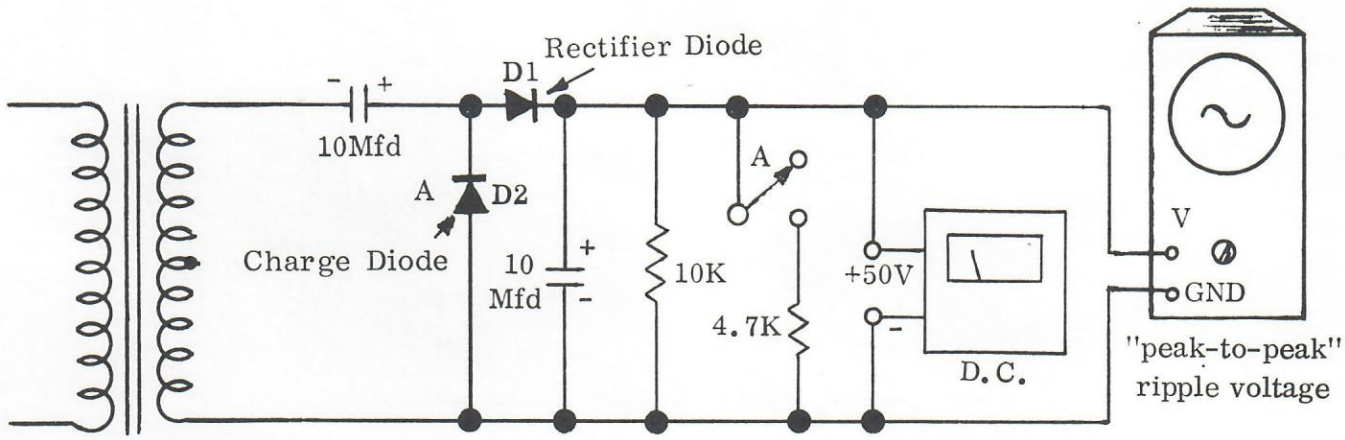



Figure 8-1 Half-Wave Voltage Doubler

Objective:

You will use a "series capacitor" which "charges-up" like a series battery thru diode A to result in a voltage doubler circuit. This circuit is important because the same method is extended to provide a voltage tripler and quadrupler.

Procedure:

1. Wire the circuit.
2. Measure the D.C. Voltage and Ripple at NO-LOAD and under LOAD. Enter the data below.

| LOAD | D.C. VOLTAGE | A.C. RIPPLE |
|---|---|---|
| NO LOAD (Switch A up) | | |
| LOAD (Switch A down) | | |

3. Calculate the percentage ripple under LOAD and the regulation.

$$\text{RIPPLE} = \frac{\text{A.C. Ripple}}{V_{\text{D.C.}}} \times 100\% \qquad \text{REGULATION} = \frac{V_{NL} - V_L}{V_L} \times 100\%$$

| RIPPLE = | REGULATION = |
|---|---|

4. How does the operation of the Half-Wave Doubler compare with the Full Wave Doubler (Experiment No. 7)?

| Ed-Lab | VOLTAGE TRIPLER | EXP. NO. 9<br>Page No. 1 |
|---|---|---|

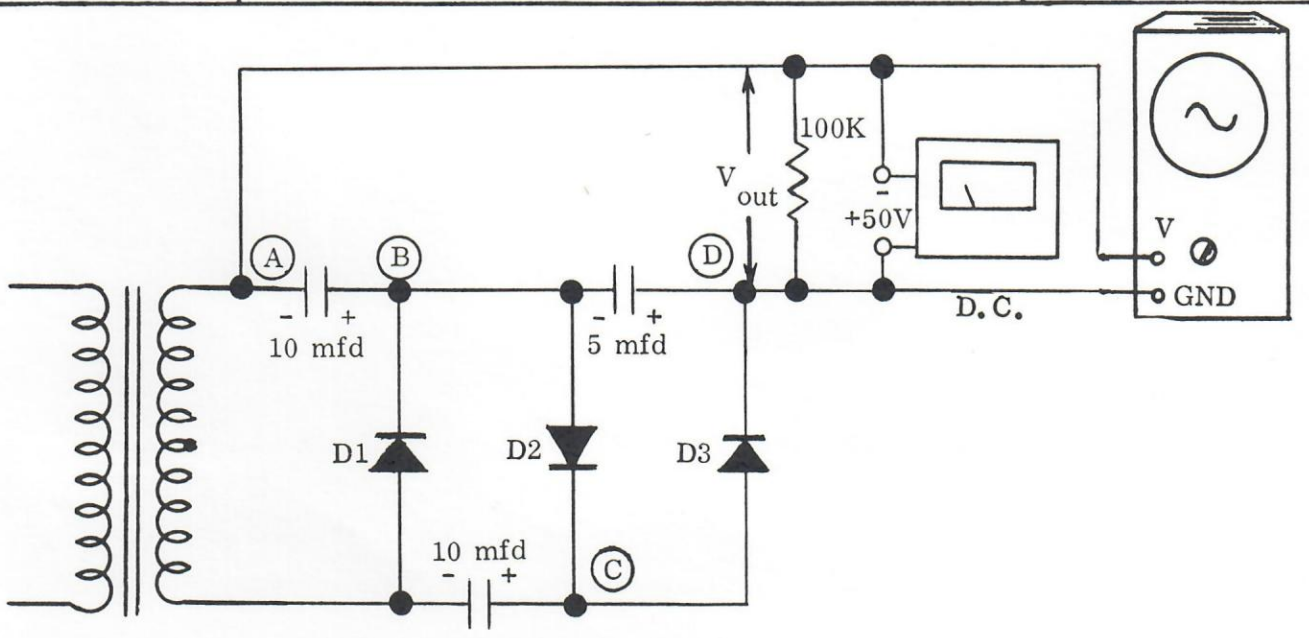



Figure 9-1 Voltage Tripler

Objective:

You will use three capacitors to provide a TRIPLER circuit which provides three times the voltage output.

Procedure:

1. Wire the circuit.
2. Measure the voltage output and ripple .

| $V_{OUTPUT}$ = | $V_{RIPPLE}$ = |
|---|---|

3. Measure the series voltage developed from the input to the output of each series capacitor: (Disconnect the D.C. Meter Leads to measure these voltages.)

| | |
|---|---|
| $V_{BA}$ = | HALF WAVE |
| $V_{CA}$ = | DOUBLER |
| $V_{DA}$ = | TRIPLER |

| Ed-Lab | VOLTAGE QUADRUPLER | EXP. NO. 10<br>Page No. 1 |
|---|---|---|

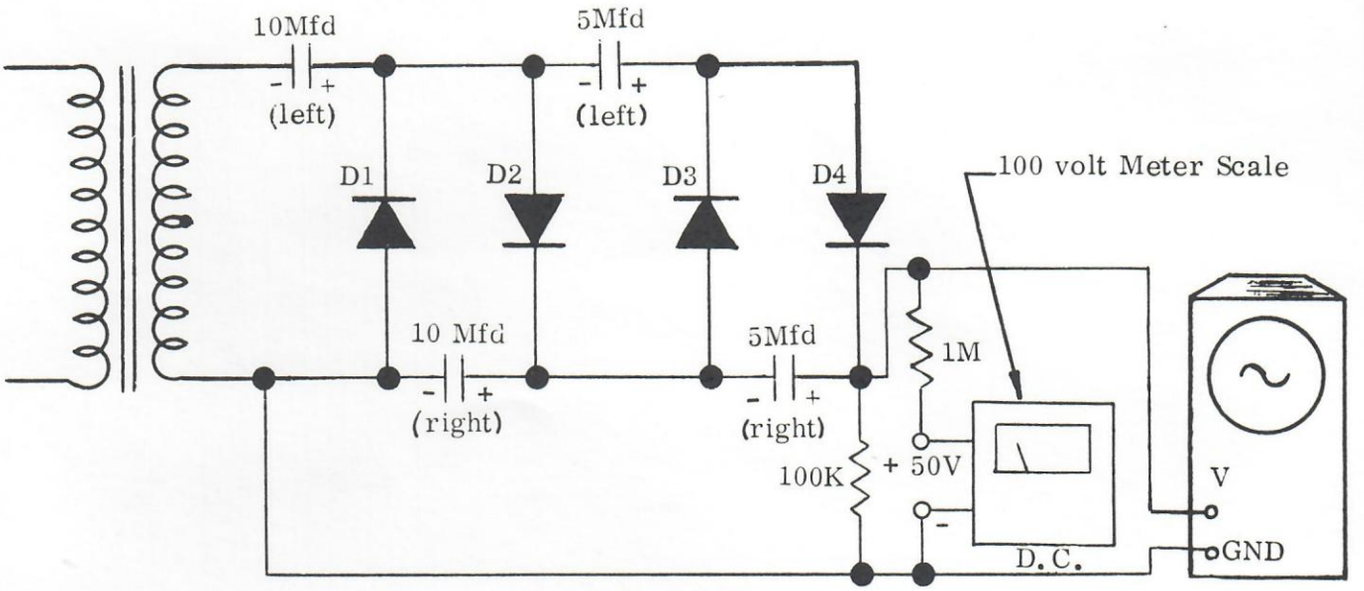



Figure 10-1 Voltage Quadrupler

Objective:

You will add another series capacitor and diode to result in a Voltage QUADRUPLER Circuit. After testing the voltage doubler and tripler you can imagine that this type of circuit can be extended to produce any size voltage multiplication. A series 1M resistor results in a meter full scale of 100 volts.

Procedure:

1. Wire the circuit and measure the voltage output and ripple.

| D.C. $V_{OUT}$ = | $V_{RIPPLE (A.C.)}$ = |
|---|---|

2. Summarize the voltage outputs from the previous circuits.

| Exp. No. | CIRCUIT | VOLTAGE OUTPUT (NO LOAD) |
|---|---|---|
| 3 | FULL WAVE (with cap) | |
| 4 | BRIDGE | |
| 8 | HALF-WAVE VOLTAGE DOUBLER | |
| 7 | FULL-WAVE VOLTAGE DOUBLER | |
| 9 | VOLTAGE TRIPLER | |
| 10 | VOLTAGE QUADRUPLER | |

| Ed-Lab | ZENER DIODE | EXP. NO. 11<br>Page No. 1 |
|---|---|---|

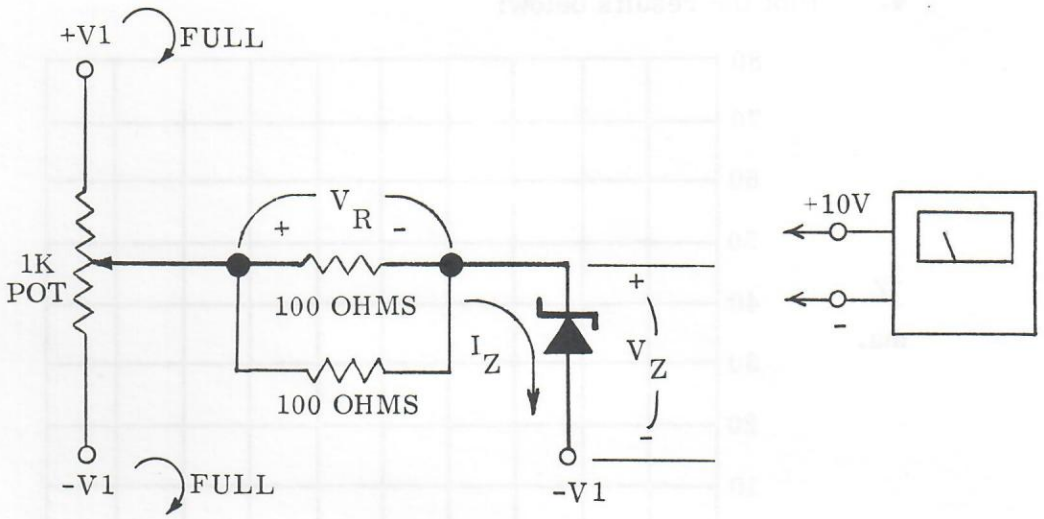



Figure 11-1 Zener Diode

Objective:

You will measure the Voltage Drop ($V_Z$) across a ZENER diode as a function of the current ($I_Z$) in the diode.

Procedure:

1. Wire the circuit.
2. Vary the input voltage with the 1K POT and measure the voltage across the ZENER ($V_Z$) and the voltage across the resistor ($V_R$). Enter the data below:

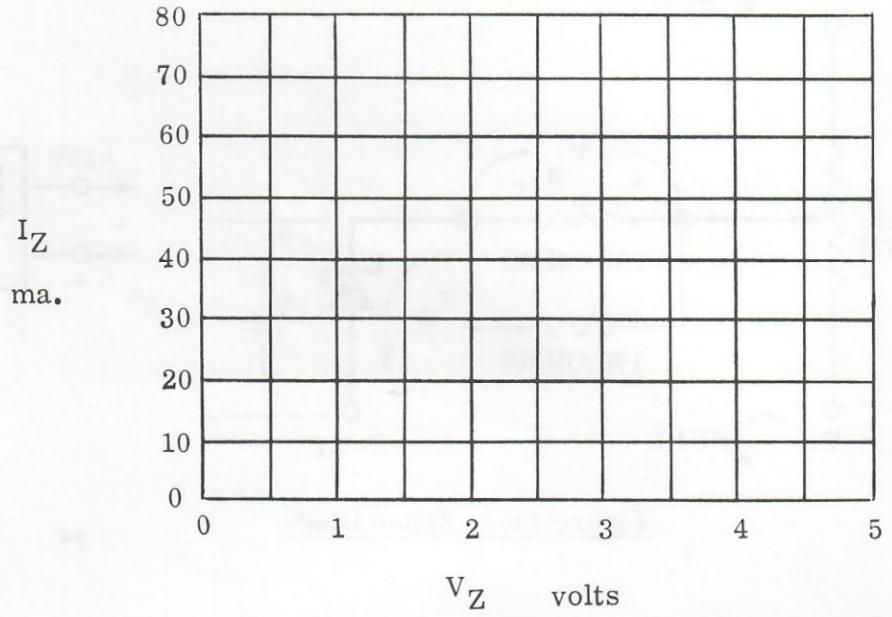
| MEASURE | | CALCULATE |
|---|---|---|
| $V_Z$ | $V_R$ | $I_Z = V_R$ x 20 ma. |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |

Example:
For $V_R$ = 6 volts
or $I_Z = V_R$ x 20 ma = 120 ma

3. Calculate the current $I_Z = V_R$ x 20 ma. Enter the results above.

| Ed-Lab | ZENER DIODE | EXP. NO. 11<br>Page No. 2 |
|---|---|---|

4. Plot the results below:




5. What is the Zener Voltage or constant voltage?

| $V_{ZENER}$ = |
|---|

6. What is the minimum Zener Current?

| $I_{ZENER}$ = |
|---|

7. What is the Zener Resistance?

This is calculated in the constant zener voltage range as

$$R_Z = \frac{\Delta V}{\Delta I} = \frac{(V_2 - V_1)}{(I_2 - I_1)}$$

$(V_2, I_2)$ and $(V, I,)$ are any two points on the "steep" curve

Use two sets of readings.

| $R_{ZENER}$ = |
|---|

| Ed-Lab | ZENER CHARACTERISTIC CURVE | EXP. NO. 12<br>Page No. 1 |
|---|---|---|

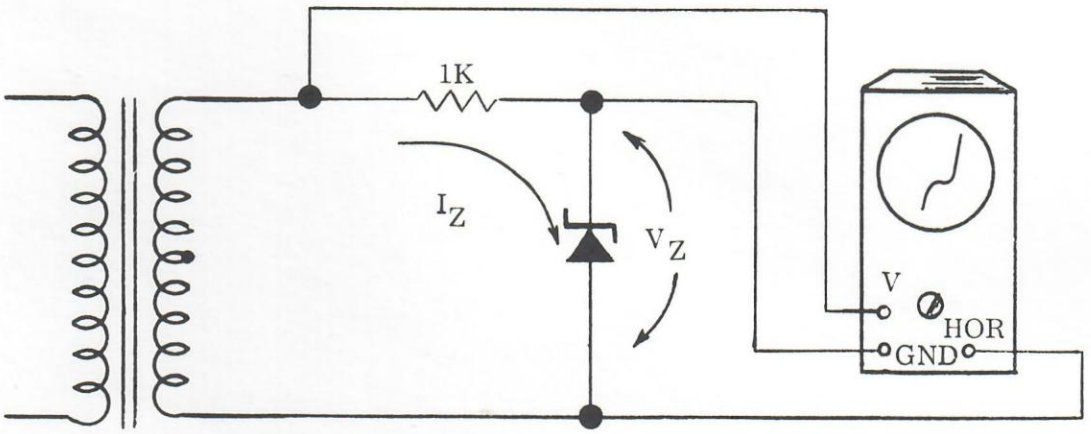


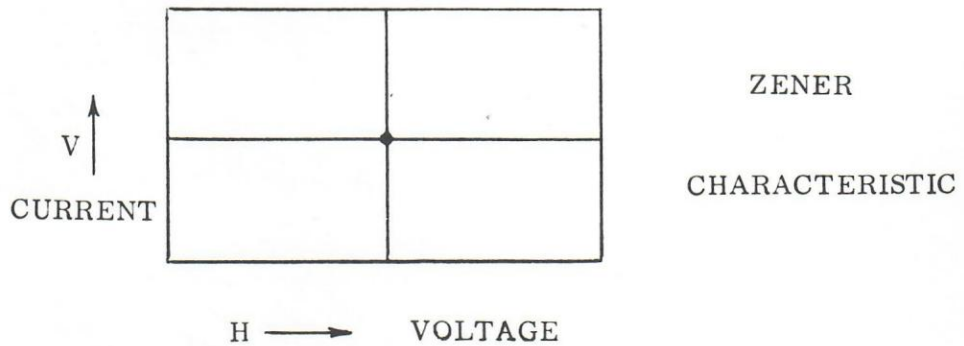

Figure 12-1 Zener Characteristic Curve

Objective:

You will use the 60 cycle transformer output to "sweep" the voltage and current and present the entire CHARACTERISTIC CURVE for the Zener diode for both positive and negative voltages.

Procedure:

1. Wire the circuit. Be sure to set the oscilloscope sweep to EXTERNAL HORIZONTAL.
2. Shut the Ed-Lab POWER to OFF and center the DOT at the middle of the scope. (Use D.C. Vertical and D.C. Horizontal Inputs).
3. With the Ed-Lab POWER ON, observe the curve and sketch the waveform from the oscilloscope. Mark the Zener region.




Curve may be to left or to right depending on whether horizontal input drives scope point left or right.

| Ed-Lab | ZENER REGULATOR | EXP. NO. 13<br>Page No. 1 |
|---|---|---|

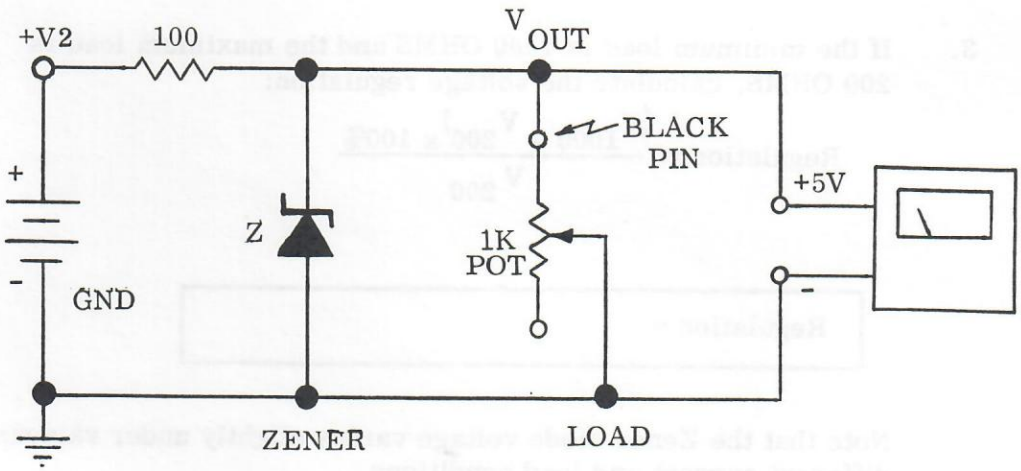



Figure 13-1 Zener Regulator

Objective:

You will use a Zener Voltage output with varying LOAD resistance and observe the output voltage variation.

Procedure:

1. Wire the circuit using the lK POT as the LOAD.
2. Vary the POT position (Load Resistance) and measure the output voltage.

   Enter the data below:

| POT POSITION | LOAD | $V_{OUT}$ |
|---|---|---|
| 1 | 100 | |
| 2 | 200 | |
| 3 | 300 | |
| 4 | 400 | |
| 5 | 500 | |
| 6 | 600 | |
| 7 | 700 | |
| 8 | 800 | |
| 9 | 900 | |
| 10 | 1000 | |

3. If the minimum load is 1000 OHMS and the maximum load is 200 OHMS, calculate the voltage regulation:

$$\text{Regulation} = \frac{(V_{1000} - V_{200}) \times 100\%}{V_{200}}$$

| Regulation = |
|---|

Note that the Zener diode voltage varies slightly under various different current and load conditions.

**Ed-Lab** | SERIES VARIABLE VOLTAGE | **EXP. NO. 14** **Page No. 1**

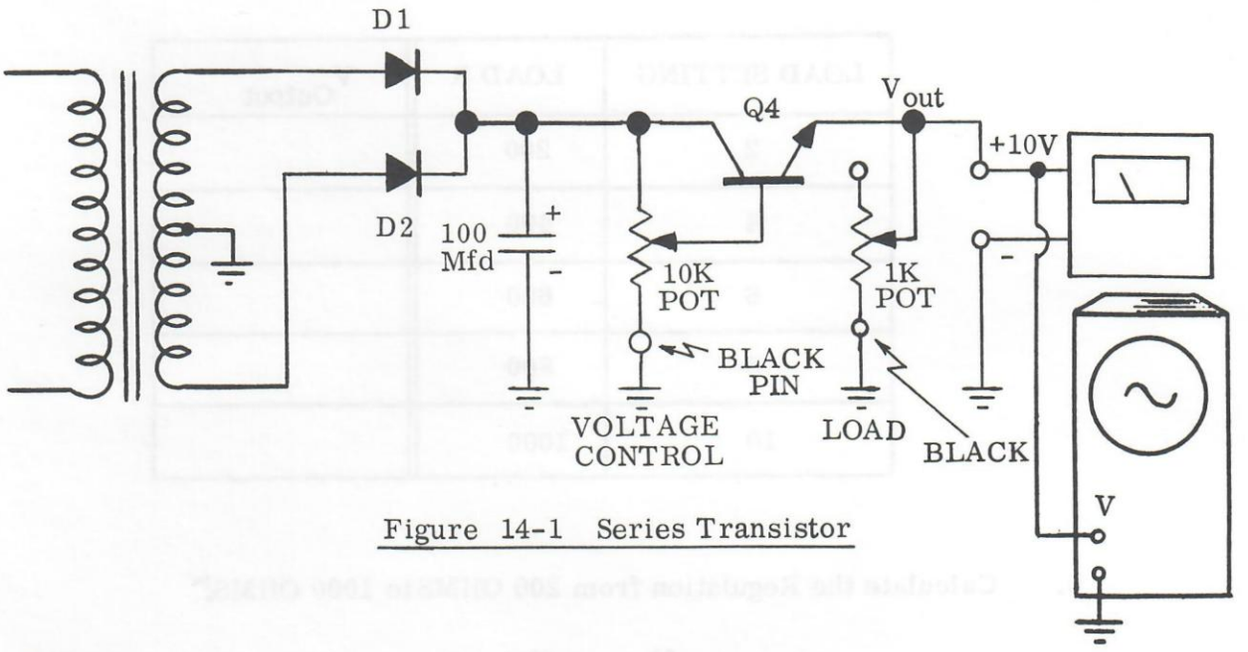



Figure 14-1 Series Transistor

Objective:

You will use a SERIES TRANSISTOR to vary the voltage output from a full wave rectifier power supply.

Procedure:

1. Wire the circuit.
2. Set the LOAD POT at about half way.
3. Vary the VOLTAGE CONTROL POT and observe the output voltage variation.
   Enter some settings below.

| CONTROL POT | OUTPUT VOLTAGE |
|---|---|
| 0 | |
| 2 | |
| 4 | |
| 8 | |
| 10 | |

4. With a mid-setting (Control POT at 4), vary the load and measure the output voltage.

   Enter the data in the table on the following page.

| LOAD SETTING | LOAD R | $V_{Output}$ |
|---|---|---|
| 2 | 200 | |
| 4 | 400 | |
| 6 | 600 | |
| 8 | 800 | |
| 10 | 1000 | |

5. Calculate the Regulation from 200 OHMS to 1000 OHMS.

$$\text{Regulation} = \frac{(V_{1000} - V_{200})}{V_{200}} \times 100\%$$

Regulation =

6. What is the Variable Voltage Range for this Power Supply?

7. Measure the Peak-to-Peak Ripple Voltage using the oscilloscope at a load of 600 ohms.

Ripple Voltage =

8. Calculate the Percentage Ripple.

Percentage Ripple =

**Ed-Lab** | SERIES REGULATOR | **EXP. NO.** 15 **Page No.** 1

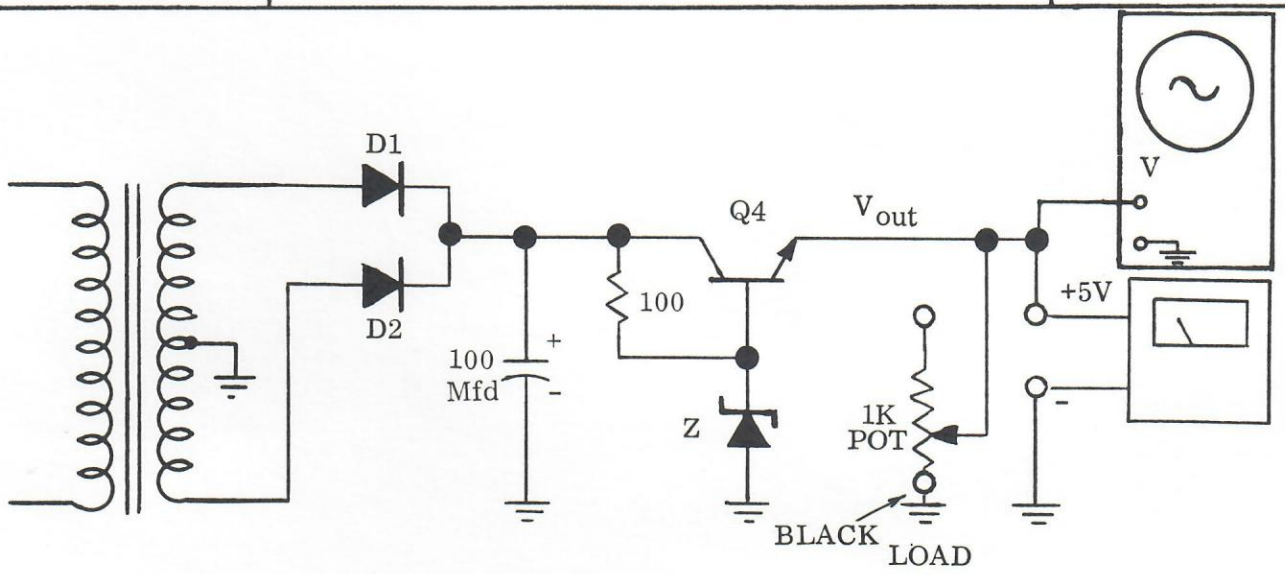



Figure 15-1 Series Transistor Regulator

Objective:

You will use a series pass transistor to control the output voltage using a Zener diode "reference".

Procedure:

1. Wire the circuit.
2. Measure the voltage output as a function of load.

| LOAD POT | LOAD | $V_{OUTPUT}$ |
|---|---|---|
| 2 | 200 | |
| 4 | 400 | |
| 6 | 600 | |
| 8 | 800 | |
| 10 | 1000 | |

3. Note that it is difficult to read changes in voltage output because the regulation is too good. The next experiment shows how to measure small changes.
4. Can you calculate the Regulation?
5. Measure the Peak to Peak Ripple Voltage and calculate the Percentage Ripple.

Percentage Ripple =

| Ed-Lab | REGULATION MEASUREMENT | EXP. NO. 16<br>Page No. 1 |
|---|---|---|

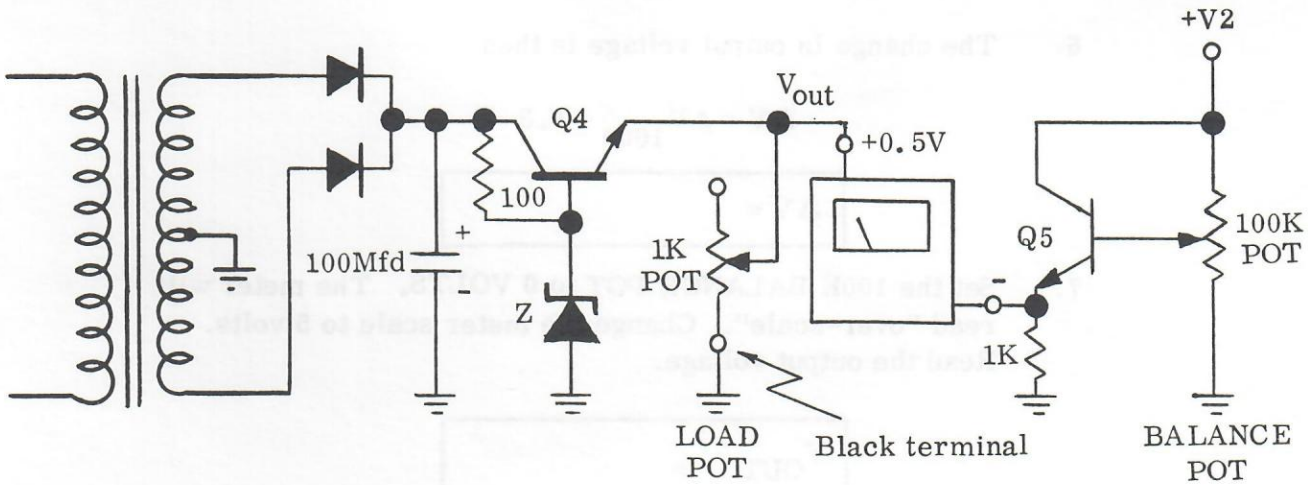



Figure 16-1 Regulation Measurement

Objective:

You will use a POTENTIOMETRIC or BALANCING measurement to detect small changes in voltages to determine REGULATION. The meter is used on a low scale (0.5 volts) and the negative terminal is set at a voltage "near" the supply output voltage to permit reading on this scale.

Procedure:

1. Wire the circuit.
2. Set the 1K LOAD POT at a load of 200 OHMS.
3. Adjust the 100K BALANCE POT so that the meter reads at some dial marking at mid-scale (0.3 volts).
4. Adjust the 1K LOAD POT to the full 1000 OHM load.
5. Read the meter.

| $V_{1000}$ = |
|---|

6. The change in output voltage is then

$$\Delta V = \Delta V_{1000} - 0.3$$

| $\Delta V =$ |
|---|

7. Set the 100K BALANCE POT to 0 VOLTS. The meter will read "over-scale". Change the meter scale to 5 volts. Read the output voltage.

| $V_{OUT} =$ |
|---|

8. Calculate the regulation:

$$R = \frac{\Delta V}{V_{out}} \times 100\%$$

| $R =$ |
|---|

| Ed-Lab | SHUNT REGULATOR | EXP. NO. 17<br>Page No. 1 |
|---|---|---|

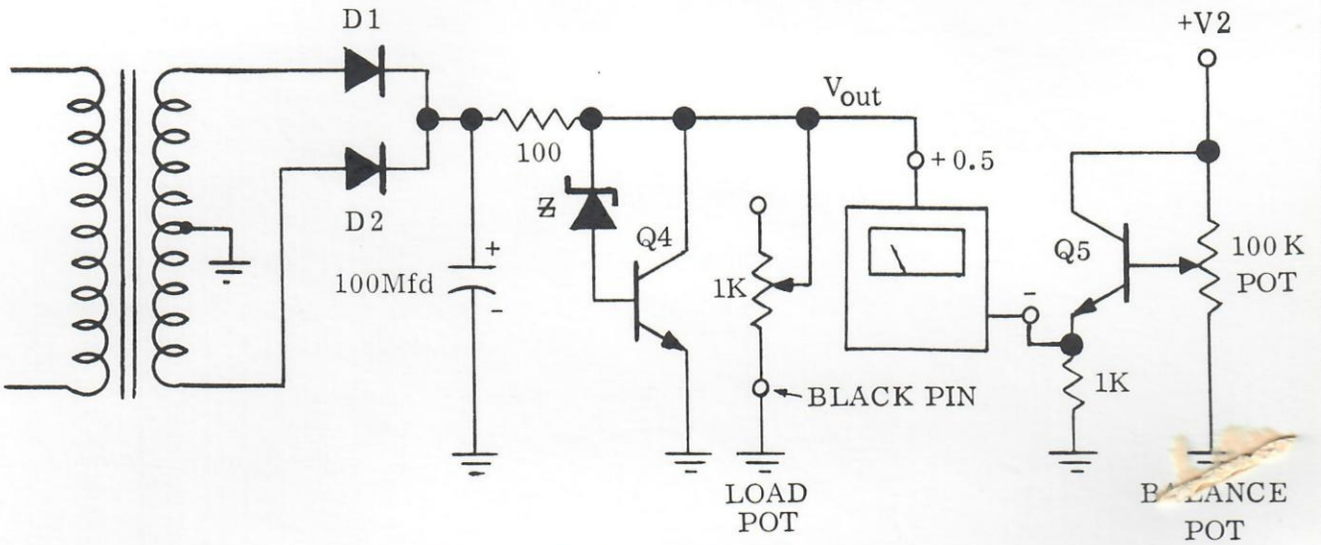



Figure 17 - 1 Shunt Regulator

Objective:

You will use a SHUNT or PARALLEL transistor to control the output voltage as a regulated power supply using the ZENER diode as a reference.

Procedure:

1. Wire the circuit.
2. Measure the output voltage at a load of 200 . The 100K BALANCE POT should be set to 0 and the +5 volt scale should be used.

| $V_{output}$ = |
|---|

3. Use the balancing method of the previous experiment to measure the voltage regulation. Use the 0.5 volt scale and adjust the 100K BALANCE POT to a mid-reading. Then read the voltage outputs at different loads.

| LOAD POT | LOAD | Relative Voltage 0.5 SCALE |
|---|---|---|
| 2 | 200 | |
| 4 | 400 | |
| 6 | 600 | |
| 8 | 800 | |
| 10 | 1000 | |

4. Calculate the Regulation

$$\text{Regulation} = \frac{(V_{1000} - V_{200})}{V_{output\ (200)}} \times 100\%$$

VOLTAGE DIFFERENCE

| R = |
|---|

| Ed-Lab | FEEDBACK REGULATION | EXP. NO. 18<br>Page No. 1 |
|---|---|---|

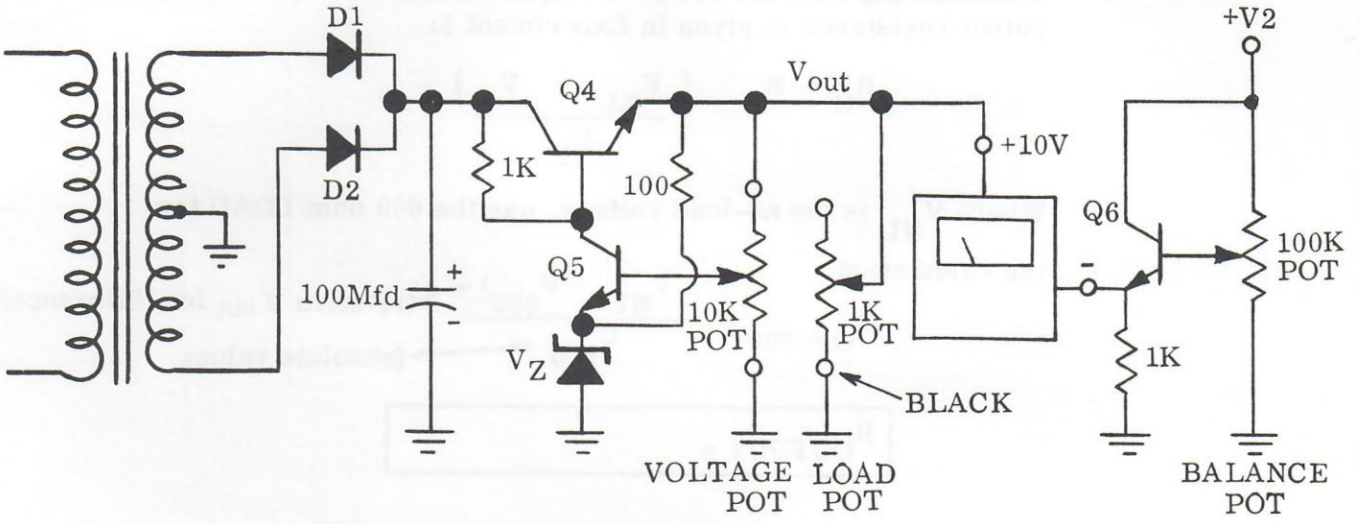



Figure 18-1 Feedback Regulation

Objective:

You will use a FEEDBACK circuit to compare the ZENER reference to the output voltage to provide an improved Regulation.

Procedure:

1. Wire the circuit and set the LOAD POT, approximately half way.

2. With the 100K Balance pot at 0, vary the Voltage Pot (10K) and observe that the output voltage varies. Enter the range variation:

| |
|---|
| $V_{maximum}$ = |
| $V_{minimum}$ = |

3. Set the output voltage for 4 volts. Connect the meter to the 0.5 volt scale and adjust the BALANCE POT for a mid-range meter reading. Vary the LOAD and enter the output voltages below:

| LOAD POT | LOAD | Relative $V_{out}$ |
|---|---|---|
| Disconnect | NO LOAD | |
| 2 | 200 | |
| 4 | 400 | |
| 6 | 600 | |
| 8 | 800 | |
| 10 | 1000 | |

4. Calculate the POWER SUPPLY output resistance. The formula for output resistance is given in Experiment 5:

$$R_O = R_L \left( \frac{V_{NL} - V_L}{V_L} \right)$$

Where $V_{NL}$ is the no-load voltage, use the 600 ohm LOAD for the calculation.

$$R_{O = 600} \left( \frac{V_{NL} - V_{600}}{V_{600}} \right)$$

($V_{NL} - V_{600}$) ← (relative $V_{out}$ for difference)

$V_{600}$ ← (absolute value)

| $R_{OUTPUT}$ = |
|---|

5. Calculate the supply regulation:

$$\text{Regulation} = \left( \frac{V_{1000} - V_{200}}{V_{200}} \right) \times 100\%$$

($V_{1000} - V_{200}$) ← (relative $V_{out}$)

$V_{200}$ ← (absolute)

| Regulation = |
|---|

6. Use the Oscilloscope to measure the voltage with a 1000 ohm load at a 5 volts output.

| $V_{ripple}$ = |
|---|

7. Calculate the percentage ripple.

| % Ripple = |
|---|

**Ed-Lab** | CURRENT REGULATOR | **EXP. NO.** 19 **Page No.** 1

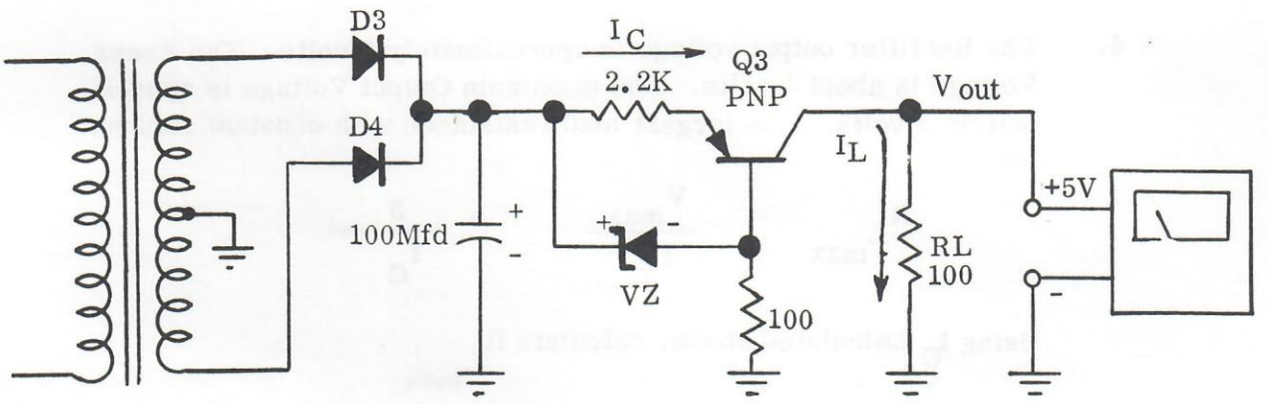



Figure 19-1 Current Regulator

Objective:

You will use a series transistor with an emitter resistor to provide a "constant current" source.

Procedure:

1. Calculate the current. (Use $V_Z$ from Experiment 11, Page 2.)

$$I_C = \frac{V_Z}{2.2K}$$ (approximately 2 ma.)

| $I_C$ (approx.) = |
|---|

2. Wire the circuit.

3. Read the output voltage for various Load Resistors ($R_L$). Enter the data below:

| $R_L$ | measure $V_{out}$ | calculate $I_L = \frac{V_{out}}{R_L}$ |
|---|---|---|
| 100 | | |
| 1K | | |
| 2.2K | | |
| 4.7K | | |
| 10K | | |
| 15K | | |
| 22K | | |
| 33K | | |
| 56K | | |

4. The Rectifier output voltage is approximately 7 volts. The Zener Voltage is about 4 volts. The maximum Output Voltage is approximately 3 volts. The largest load resistance with constant current is:

$$R_{L_{max}} = \frac{V_{max}}{I_C} = \frac{3}{I_C}$$

Using $I_C$ calculated above, calculate $R_{L_{max}}$:

$R_{L_{max}} =$

5. How does this compare with the experiment results?

**Ed-Lab** | OP - AMP REGULATOR | **EXP. NO.** 20 **Page No.** 1

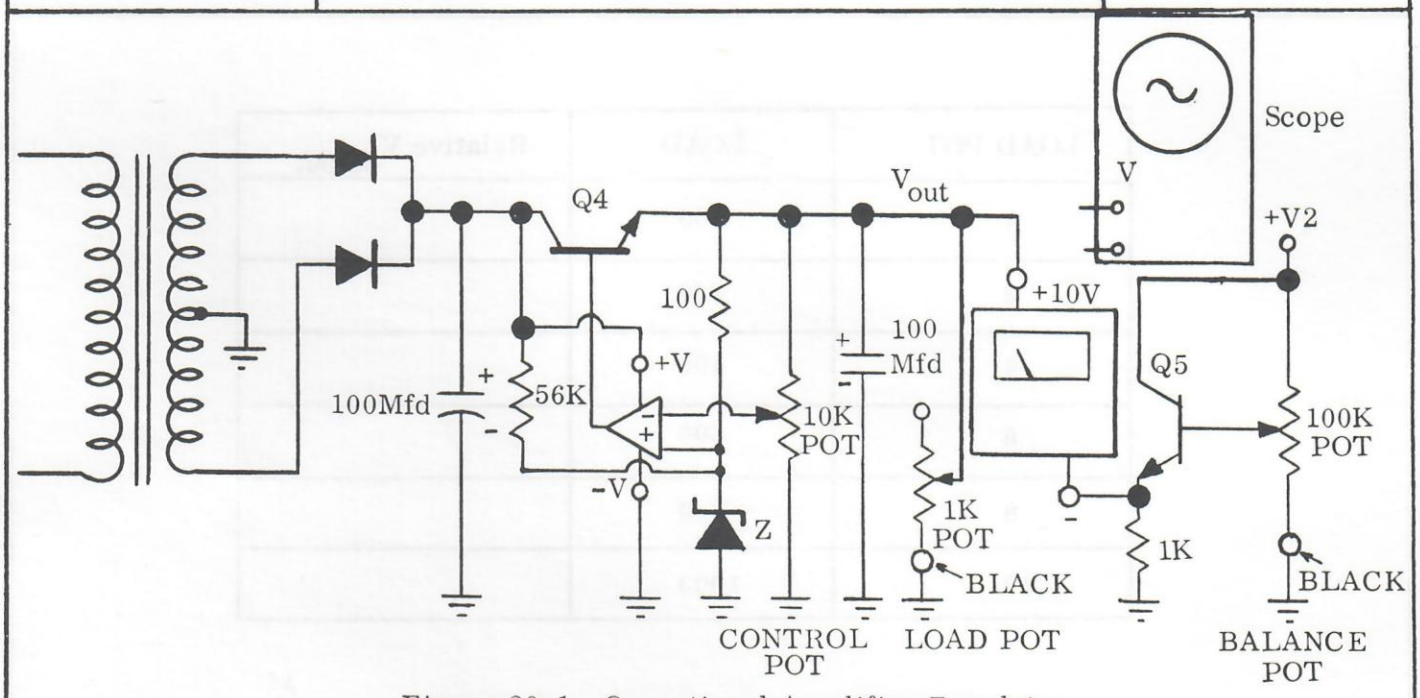



Figure 20-1 Operational Amplifier Regulator

Objective:

You will use an OPERATIONAL AMPLIFIER in a FEEDBACK loop to Regulate the Voltage.

Procedure:

1. Wire the circuit with the LOAD POT approximately half-way and the BALANCE POT to 0.

2. Vary the control POT and enter the maximum and minimum voltage below:

| |
|---|
| $V_{max}$ = |
| $V_{min}$ = |

3. Set the output voltage (10K POT) to 4 volts. Use the 0.5 volt scale and BALANCE (100K POT) the meter to read mid-scale. Vary the LOAD (1K POT) and enter the output voltages in the table on the following page.

| Ed-Lab | OP - AMP REGULATOR | EXP. NO. 20<br>Page No. 2 |
|---|---|---|

| LOAD POT | LOAD | Relative $V_{output}$ |
|---|---|---|
| 1 | 100 | |
| 2 | 200 | |
| 4 | 400 | |
| 6 | 600 | |
| 8 | 800 | |
| 10 | 1000 | |

4. Calculate the regulation if you were able to read a difference in voltage. If you cannot read any relative difference, estimate the smallest reading you could make on the meter (about .005 volts). Then calculate the regulation based on this value and state that the regulation is "better than":

$$\text{Regulation} = \frac{(V_{1000} - V_{200})}{V_{200}} \times 100\%$$

($V_{1000} - V_{200}$): (relative)

$V_{200}$: (absolute; 4 volts)

| Regulation = (Better than) |
|---|

5. Use the Oscillscope to measure the ripple voltage with a 1000 ohm load at 4 volts output. If you cannot obtain a reading, state that the ripple is "better than". . . .

| $V_{ripple}$ = |
|---|

6. Summarize the Regulation for the different power supplies:

| POWER SUPPLY | EXPERIMENT | REGULATION % |
|---|---|---|
| Full Wave Supply (200 OHM LOAD) | 5 | |
| Zener Regulator | 13 | |
| Series Transistor | 14 | |
| Series Zener Regulator | 16 | |
| Shunt Regulator | 17 | |
| Feedback Regulator | 18 | |
| OP - AMP Regulator | 20 | |

| Ed-Lab | POWER SUPPLY EFFICIENCY | EXP. NO. 21<br>Page No. 1 |
|---|---|---|

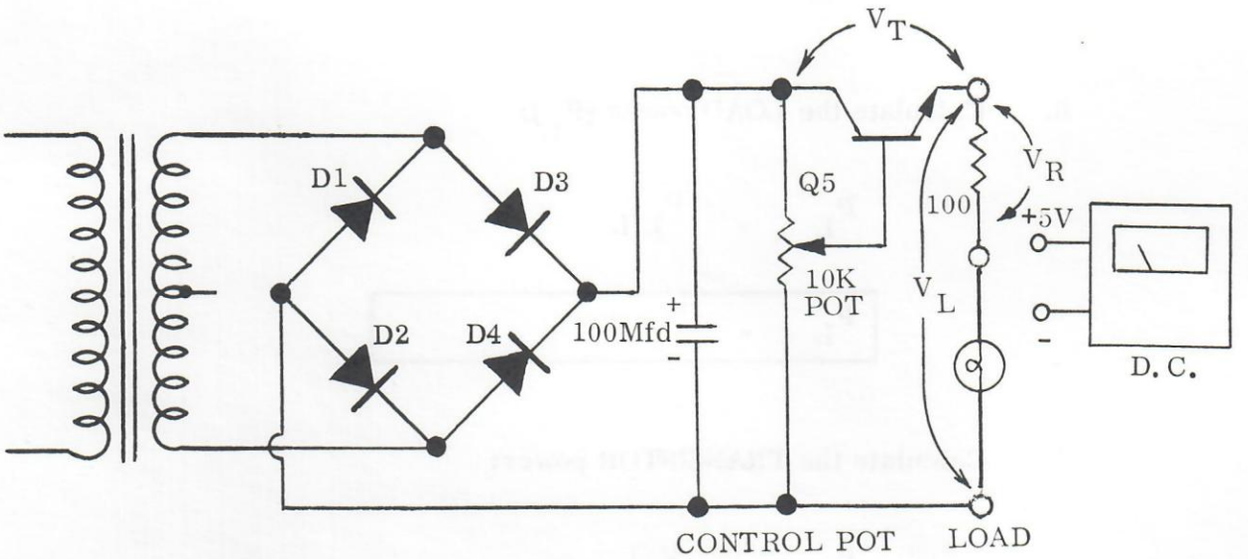



Figure 21-1 Lamp Power Supply

Objective:

You will test a FULL-WAVE Power Supply which drives a LAMP and RESISTOR LOAD from OFF and DIM to BRIGHT and measure the EFFICIENCY of the supply.

Procedure:

1. Wire the circuit.
2. Vary the 10K POT and observe that the lamp varies from bright to dim.
3. Set the CONTROL POT to mid-scale.
4. Measure the voltages:

| | |
|---|---|
| $V_T$ | = |
| $V_R$ | = |
| $V_L$ | = |

5. Calculate the LOAD current ($I_L$):

$$I_L = \frac{V_R}{100}$$

| | |
|---|---|
| $I_L$ | = |

6. Calculate the LOAD power ($P_L$):

$$P_L = V_L I_L$$

| $P_L$ = |
|---|

7. Calculate the TRANSISTOR power:

$$P_T = V_T I_L$$

| $P_T$ = |
|---|

8. Calculate the efficiency. The Efficiency of the Regulation is the output power ($P_L$) divided by the total power ($P_L + P_T$) (Load + Transistor Regulator).

$$E = \frac{P_L}{(P_L + P_T)} \times 100\%$$

| E = |
|---|

This efficiency does not include the power "loss" in the rectifier diodes or in the transformer. If we were to measure the total A.C. Power input, then we would determine the total power supply efficiency.

**Ed-Lab** | COLLECTOR OUTPUT POWER | **EXP. NO.** 22 **Page No.** 1

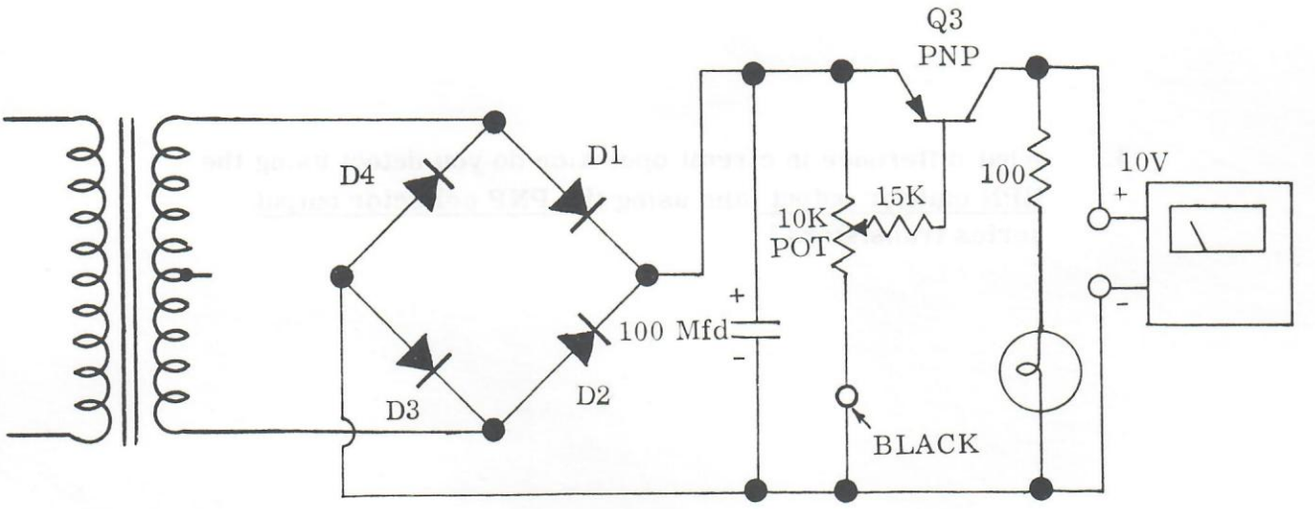



Figure 22-1 PNP Collector Output Power Supply

Objective:

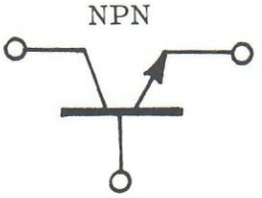
You will test a FULL WAVE power supply which drives a LAMP and RESISTOR LOAD using a series PNP transistor with the output from the COLLECTOR.

Procedure:

1. Wire the circuit.
2. Vary the 10K POT and observe the variation in LAMP INTENSITY.
3. Vary the POT position and measure the output voltage. Enter the DATA below. Then repeat the measurements using the NPN transistor circuit from the previous experiment:

PNP

NPN



| POT POSITION | PNP transistor and 15K resistor | NPN transistor NO BASE resistor |
|---|---|---|
| 0 | | |
| 1 | | |
| 2 | | |
| 3 | | |
| 4 | | |
| 5 | | |
| 6 | | |
| 7 | | |
| 8 | | |
| 9 | | |
| 10 | | |

4. What difference in circuit operation do you detect using the NPN emitter output and using the PNP collector output series transistors?

| Ed-Lab | DARLINGTON SERIES REGULATOR | EXP. NO. 23<br>Page No. 1 |
|---|---|---|

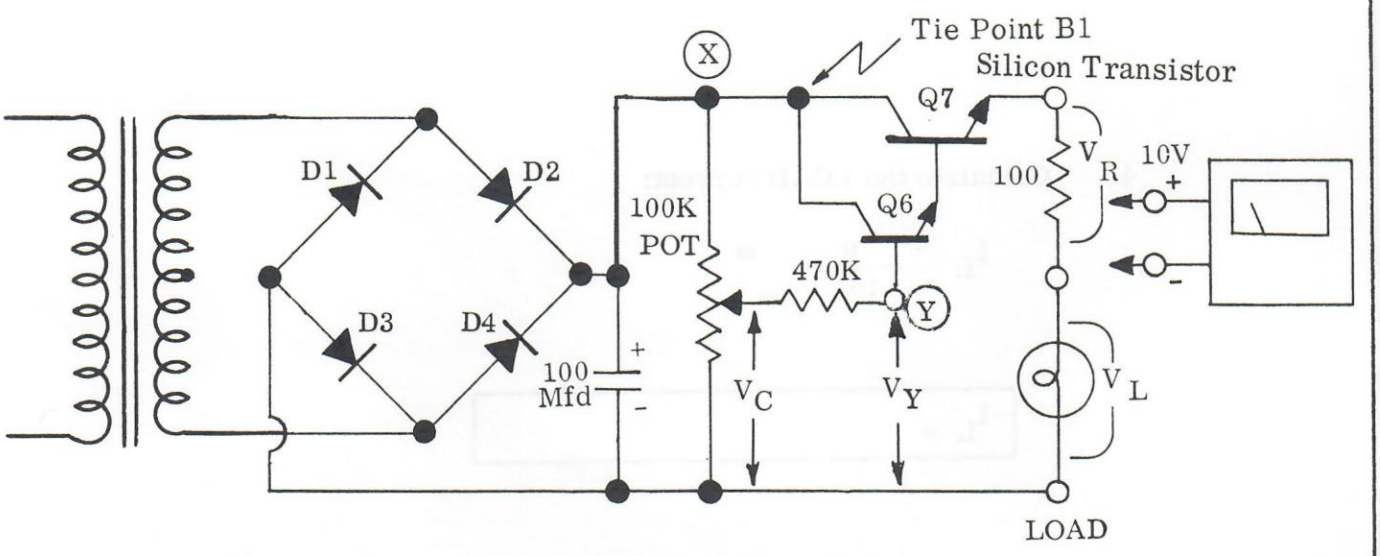



Figure 23-1 Darlington Supply

Objective:

You will use a series transistor with a DARLINGTON driving transistor. The base resistor used is 470K providing very little driving current. This circuit is used in very high current power supplies.

Procedure:

1. Wire the circuit.
2. Vary the 100K CONTROL POT, and observe the variation in Lamp Intensity.
3. With the Lamp Full-ON, 100K POT in Position 8, measure the following voltages:

| | |
|---|---|
| $V_R$ | |
| $V_L$ | |
| $V_Y$ | |
| $V_C$ | |

4. Calculate the LOAD current:

$$I_L = \frac{V_R}{100} =$$

| $I_L$ = |
|---|

5. Calculate the "Transistor BASE" DRIVING current:

$$I_D = \frac{(V_Y - V_C)}{470K}$$

| $I_D$ = |
|---|

6. Calculate the CURRENT GAIN for the "Darlington Pair".

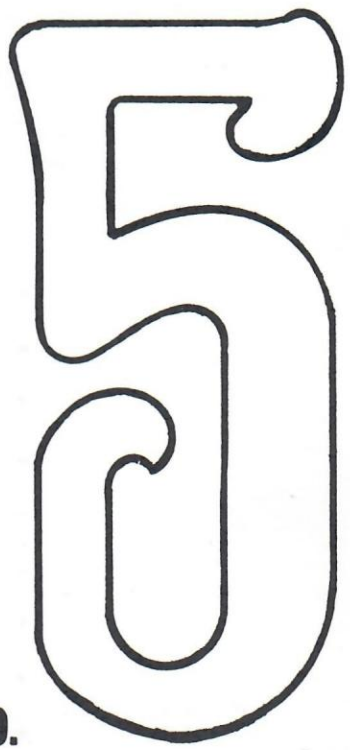
$$G = \frac{I_L}{I_D}$$

| G = |
|---|

# Ed-Lab

## *EXPERIMENT MANUAL*

**INDUSTRIAL ELECTRONICS CONTROLS**



**UNIT NO.**

**CES INDUSTRIES, INC.**

**Ed-Lab** | UNIT NO. 5

# EXPERIMENT MANUAL

# INDUSTRIAL ELECTRONIC CONTROLS

Copyright © 1974, 1976, 1980, 1981 CES INDUSTRIES, INC.

Name _____

Course _____

TABLE OF CONTENTS


1. Transistor Switch
2. PNPN Switch
3. SCR Solid-State Switch
4. SCR Rectifier Control Switch
5. R-C Phase Shift
6. Phase Shift SCR Control
7. SCR Static Control
8. Unijunction Transistor
9. Unijunction Sweep
10. Unijunction SCR Control
11. Unijunction Delay-ON Control
12. DIAC Operation
13. DIAC Characteristic Curve
14. TRIAC Operation
15. TRIAC Phase Shift Control
16. TRIAC Latching & Holding Currents
17. Digital Astable Multivibrator
18. Digital Series Voltage Control
19. Photocell SCR Control
20. Step-up Transformer *
21. Auto-Transformer *
22. Ringing-Choke Converter/Inverter *
23. D.C. to A.C. Converter *
24. D.C. to D.C. Converter *
25. Two-Transformer Inverter *
26. Transformer Inductance *
27. A.C. Magnetic Amplifier *
28. D.C. Magnetic Amplifier *
29. Full Wave SCR Bridge *


* Requires External Transformer Package

| Ed-Lab | TRANSISTOR SWITCH | EXP. NO. 1<br>Page No. 1 |
|---|---|---|

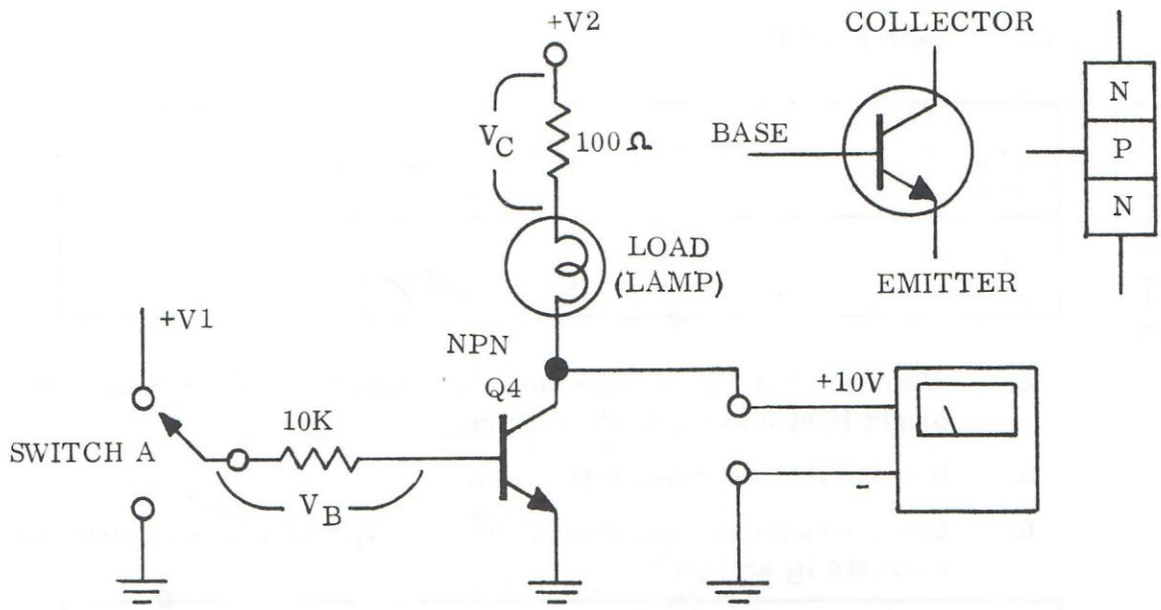



Figure 1-1 Transistor Switch

Objective:

You will use a mechanical switch (A) to control a small current which energizes a Solid-State TRANSISTOR switch to control a LAMP used as a LOAD.

Procedure:

1. Wire the circuit from Figure 1-1. The LAMP is located at Tie Point B7. Set +V1 to maximum Clockwise.

2. Operate the Switch UP and DOWN to control the current in the lamp. Read the switch "collector" voltage when the switch is "open" and "closed" and enter the data below:

| SWITCH A | TRANSISTOR SWITCH | COLLECTOR VOLTAGE | LAMP (ON OR OFF) |
|---|---|---|---|
| UP | CLOSED | | |
| DOWN | OPEN | | |

3. The lamp is ON with Switch A UP. With Switch A UP, use the 2 leads from the Ed-Lab METER and measure the voltages $V_B$ and $V_C$. Then calculate the input current $\left(\frac{V_B}{10K}\right)$ and the load current $\left(\frac{V_C}{100}\right)$

Switch A UP

| | |
|---|---|
| $V_B$ = | $I_B = \frac{V_B}{10,000}$ = |
| $V_C$ = | $I_C = \frac{V_C}{100}$ = |

4. With Switch A UP, reduce the +V1 voltage to result in a medium bright light lower than the maximum.

5. Remove the two meter test wires.

6. Use the meter and two wires, measure $V_B$ and $V_C$. Calculate the currents $I_B$ and $I_C$:

| | |
|---|---|
| $V_B$ = | $I_B$ = |
| $V_C$ = | $I_C$ = |

7. Calculate the current gain of the transistor.

| |
|---|
| CURRENT GAIN = $\frac{I_C}{I_B}$ = |

| Ed-Lab | PNPN SWITCH | EXP. NO. 2<br>Page No. 1 |
|---|---|---|

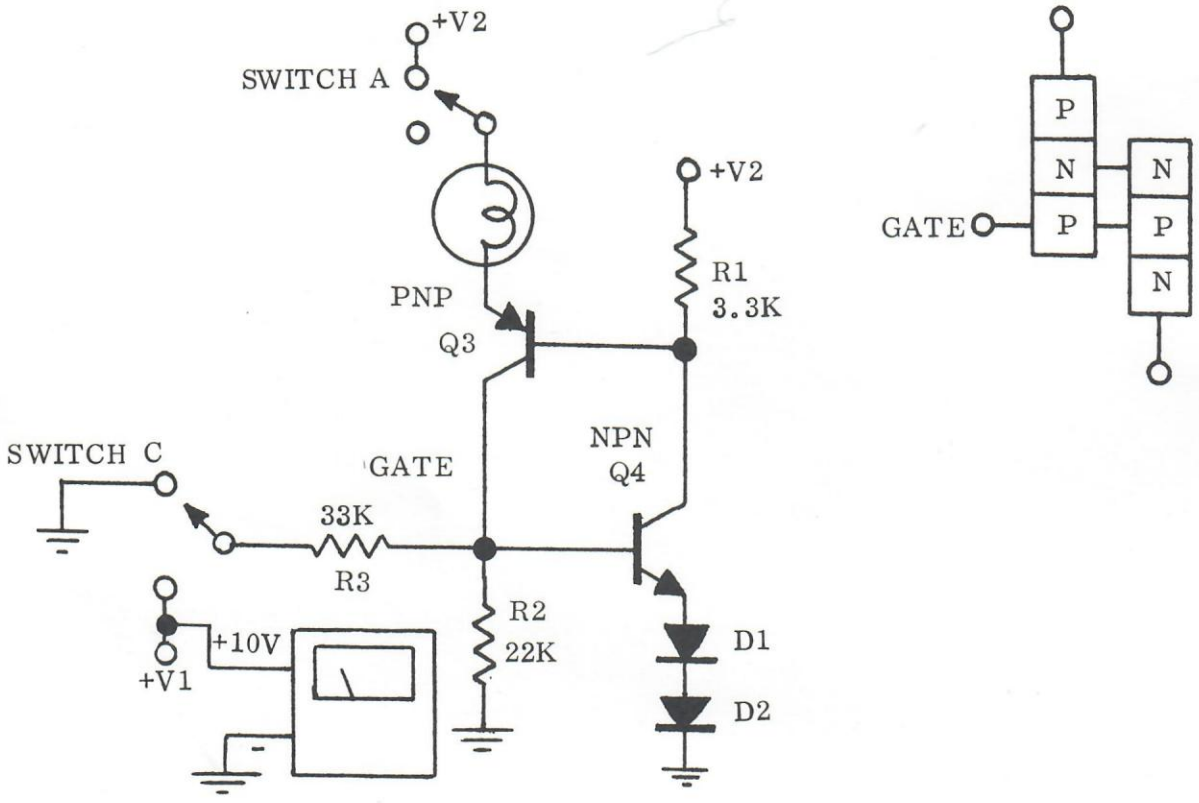



Figure 2-1 PNP-NPN Switch

Objective:

You will use a PNP and an NPN Transistor in a "feedback" circuit with each collector (output) connected to the other transistor base (input). The transistors can both be OFF, or both can be ON and the lamp will be ON.

Procedure:

1. Wire the circuit. Normally, Silicon transistors are used because of their lower "leakage" and base "threshold" which is higher than that of germanium. In the above circuit, Diodes D1 and D2 and Resistor R1 and R2 are included to prevent any leakage from "turning-on" the transistors when there is no input GATE voltage.
2. Switch A and C should be UP.
3. Set Switch A DOWN, then UP. The Lamp should now be OFF.
4. Set Switch C DOWN (Input) (to +V volts)
   The Lamp should now be ON.
5. Set Switch C UP.
   The Lamp should remain ON. The two transistors hold or LATCH on.
6. The Lamp can again be shut-OFF with Switch A DOWN, then UP again. This removes current and unlatches the switch.
7. Does the circuit operate as a "latching" switch?

| Ed-Lab | SCR SOLID-STATE SWITCH | EXP. NO. 3<br>Page No. 1 |
|---|---|---|

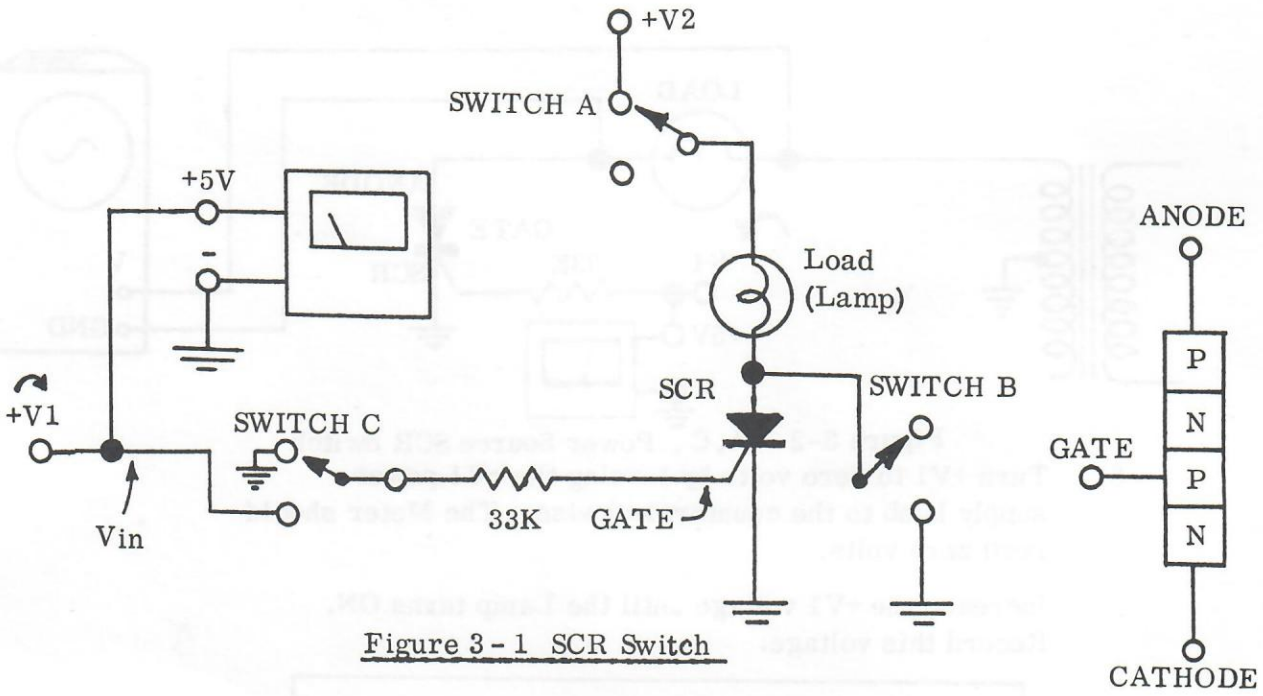



Figure 3 - 1 SCR Switch

Objective:

You will use an SCR three terminal semiconductor which is actually a PNPN device as a SOLID-STATE SWITCH. Two methods are used to shut the Switch OFF. The gate TURN-ON voltage is measured.

Procedure:

1. Wire the circuit in Figure 3 - 1 with all switches UP. Set +V1 to maximum.
2. Set Switch C DOWN, then UP. Does the lamp turn ON and remain ON?
3. Set Switch A DOWN, then UP. Does the lamp turn OFF?
4. Set Switch C DOWN, then UP to turn the lamp ON. Set Switch B DOWN, then UP. Does the lamp turn OFF?
5. Turn +V1 to zero volts by turn the +V1 power supply knob counterclockwise. The Meter should read zero volts.
6. Move switch C DOWN and increase the voltage until turn-on:

   $V_{in}$ (at turn-on) =

   Move switch C UP and observe that the Lamp Remains ON.
7. The same circuit operates with an AC source where the SCR results in a Rectified D.C. Load current. Wire the circuit in Figure 3 - 2.

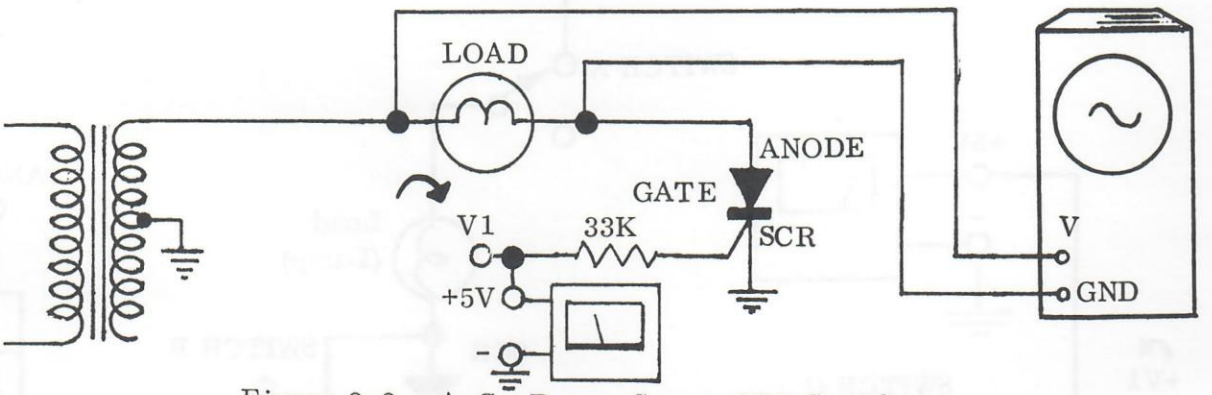



Figure 3-2 A.C. Power Source SCR Switch

8. Turn +V1 to zero volts by turning the +V1 power supply knob to the counterclockwise. The Meter should read zero volts.

   Increase the +V1 voltage until the Lamp turns ON. Record this voltage:

   $V_{in}$ (TURN-ON) =

9. Sketch the oscilloscope waveform across the LOAD when the Lamps is ON.

This waveform is the same as that for a "half-wave rectifier". The Gate control voltage turns-on the SCR at each cycle. Unlike the D.C. Circuit in Figure 3 - 1 when the Gate Voltage is lowered, the LOAD current is turned-off.

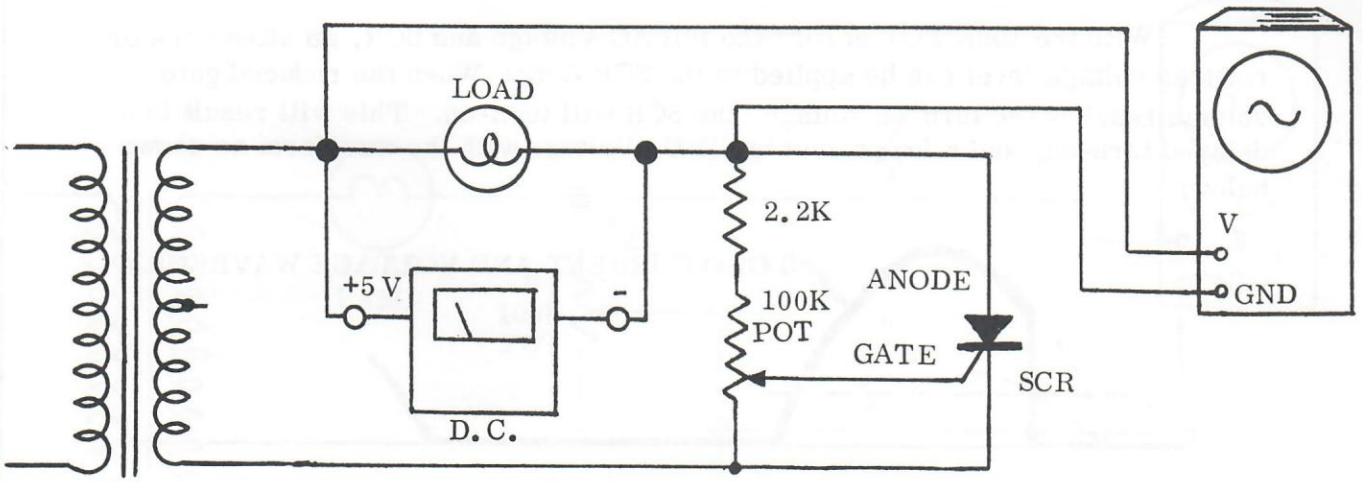



Figure 4-1 SCR Silicon Controlled Rectifier

Objective:

You will use an SCR and an AC Voltage Source with a Potentiometer to reduce the voltage to adjust the turn-on time. The load voltage waveform is observed on the Oscilloscope.

Description:

The A.C. Voltage across the SCR varies from 0 (zero volts) to full positive and full negative volts, as shown below:

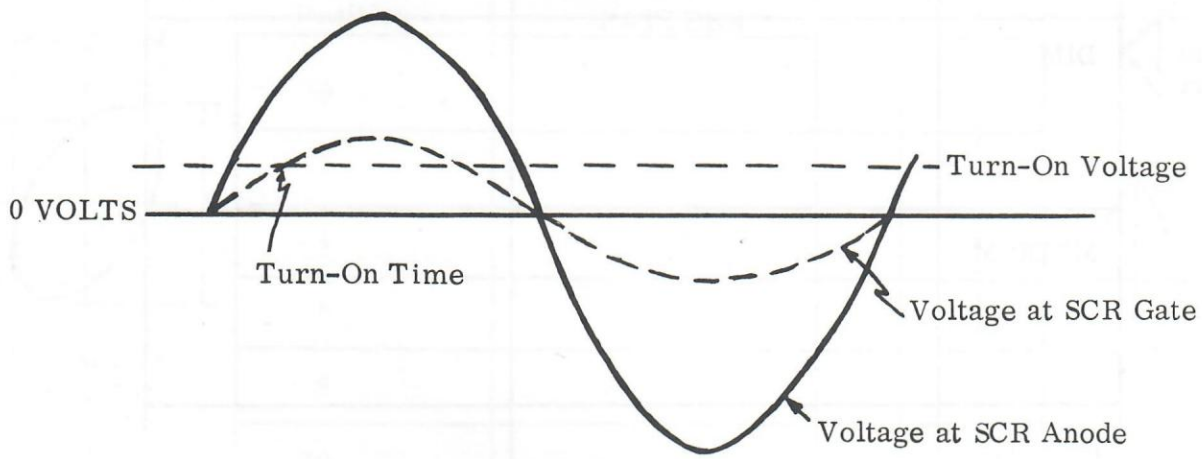



Figure 4-2 A.C. Voltage and Reduced Gate Voltage

With the 100K POT across the full AC Voltage and SCR, an attenuated or reduced voltage level can be applied to the SCR Gate. When the reduced gate voltage reaches the turn-on voltage, the SCR will turn-on. This will result in a delayed turn-on, and a lower average D.C. Voltage with the waveform as shown below:

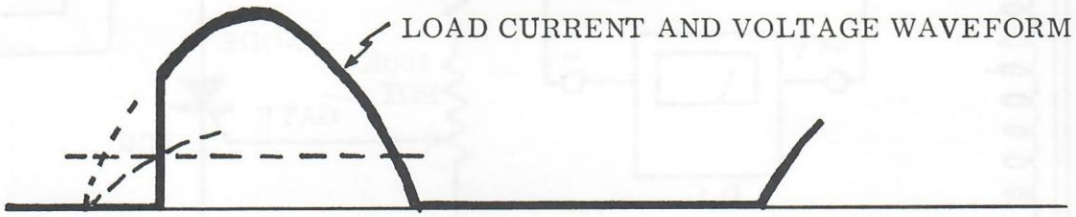



Figure 4-3 SCR Delayed Turn-On Waveform

When the Gate Voltage is reduced where the peak reaches the turn-on volts, this provides the maximum delay. Any further reduction will lower the gate voltage where it does not reach the turn-on voltage.

Procedure:

1. Wire the circuit shown in Figure 4-1.
2. Vary the "gate voltage" using the 100K POT. Observe the waveform and the D.C. voltage readings. Enter the data below:

| LAMP | WAVEFORM | D.C. Volts (Average) |
|---|---|---|
| DIM | | |
| MEDIUM | | |
| BRIGHT | | |

**Ed-Lab** | R - C PHASE SHIFT | **EXP. NO.** 5 **Page No.** 1

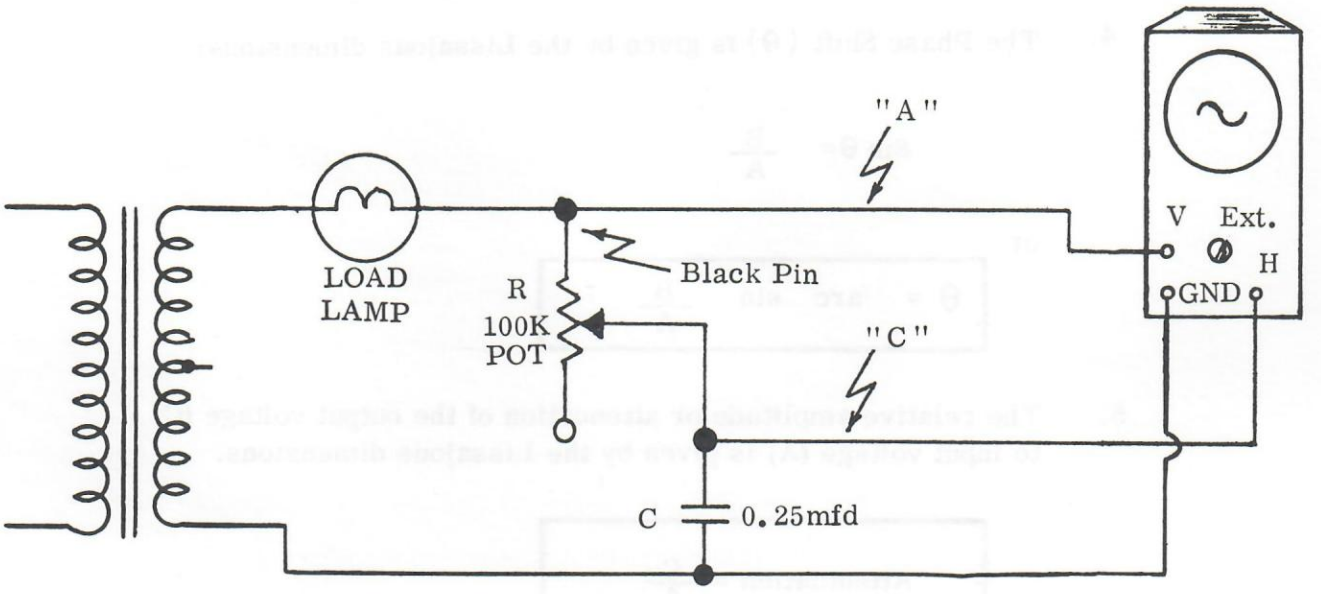


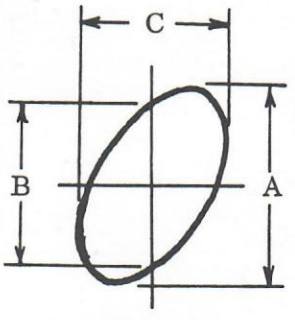

Figure 5-1 R-C Phase Shift Circuit

Objective:

You will use a Resistor-Capacitor Series network to provide a DELAY or PHASE SHIFT. A Lissajous Pattern on an Oscilloscope will be used to observe the Phase Shift and Amplitude Variation.

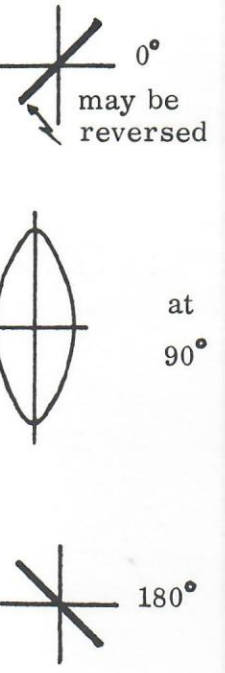
Procedure:

1. Wire the circuit shown in Figure 5-1.
2. Vary the 100K Pot and observe the output Lissajous Patterns. Plot the waveforms below and read the A, B, and C dimensions.




LISSAJOUS PATTERN

| 100K POT Position | LISSAJOUS PATTERN | Lissajous A | B | C |
|---|---|---|---|---|
| 0 | | | | |
| 2 | | | | |
| 4 | | | | |
| 6 | | | | |
| 8 | | | | |
| 10 | | | | |




3. If a Dual-Trace Scope is available, use CHOPPED SWEEP and view waveforms at A and B, and determine phase shift.

4. The Phase Shift ( $\theta$) is given by the Lissajous dimensions:

$$\sin \theta = \frac{B}{A}$$

or

$$\theta = \text{arc} \quad \sin \quad \frac{B}{A}$$

5. The relative amplitude or attenuation of the output voltage (C) to input voltage (A) is given by the Lissajous dimensions.

$$\text{Attentuation} = \frac{C}{A}$$

6. What are the phase- shift and the attenuation at POT setting 0?

7. What are the maximum phase-shift and maximum attenuation at POT setting 10?

**Ed-Lab** | PHASE SHIFT SCR CONTROL | **EXP. NO.** 6 **Page No.** 1

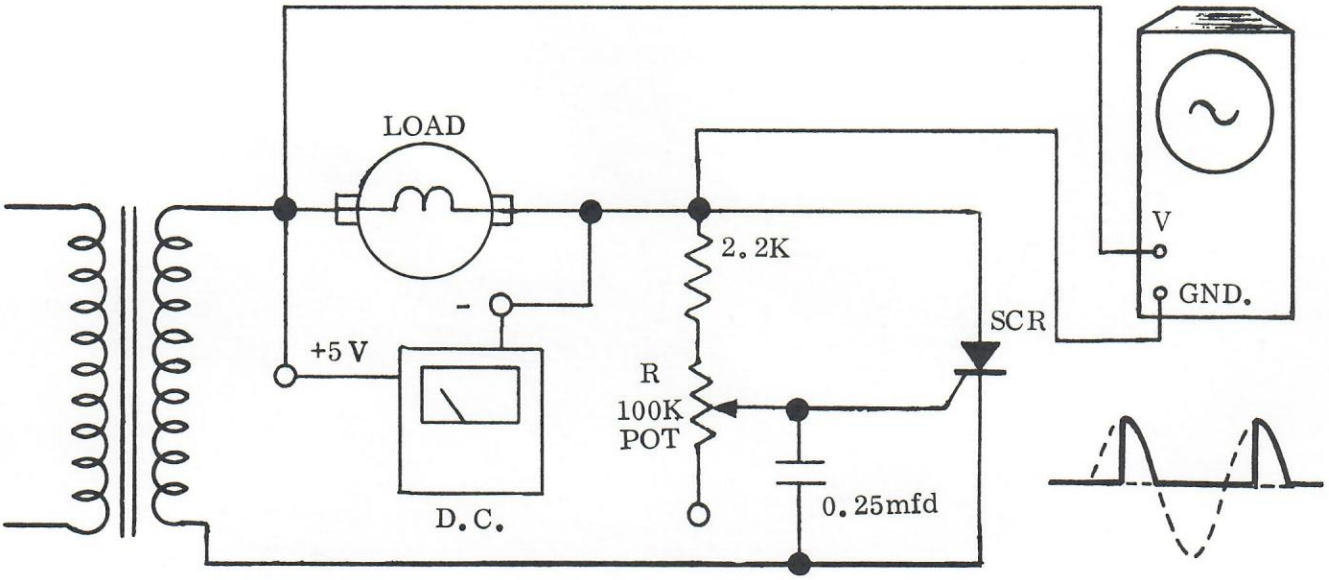



Figure 6-1 Phase Shift Control

Objective:

You will use a Phase Shift (Delay) network to delay the turn-on of the SCR.

Procedure:

1. Wire the circuit in Figure 6-1.
2. Vary the 100K POT and observe the waveform on the oscilloscope, and read the D.C. Voltage on the meter.

| LAMP | WAVEFORM | D.C. VOLTS |
|---|---|---|
| DIM | | |
| MEDIUM | | |
| BRIGHT | | |

3. What is the range of D.C. Voltages for the LOAD?

| $V_{max}$ = | $V_{Min}$ = |
|---|---|

4. Place another 0.25 mfd cap in parallel with C. What is $V_{min}$?
5. Replace the 100K POT with a 1M POT. What is $V_{min}$?

| Ed-Lab | SCR STATIC CONTROL | EXP. NO. 7<br>Page No. 1 |
|---|---|---|

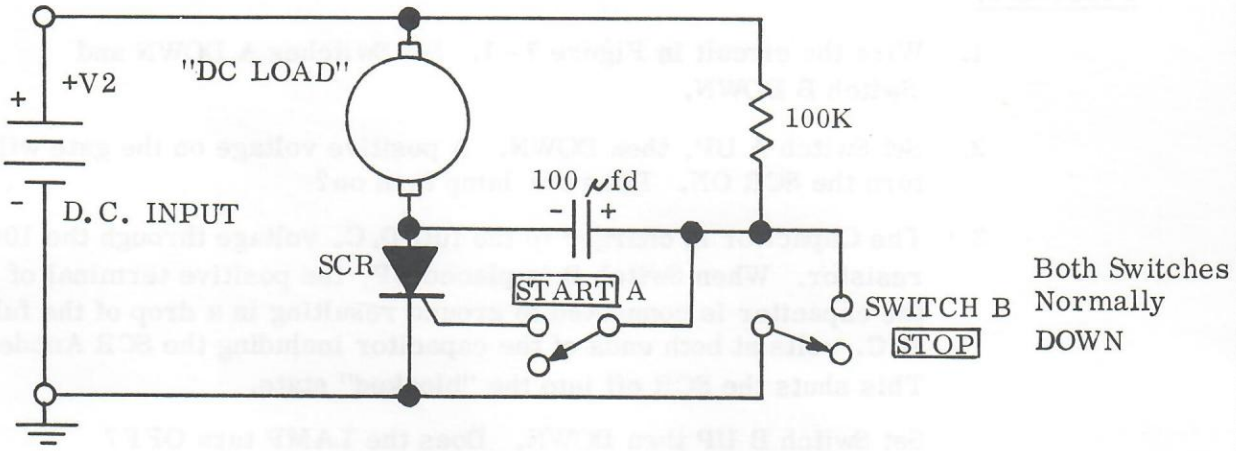



Figure 7 - 1 SCR D.C. Static Switch

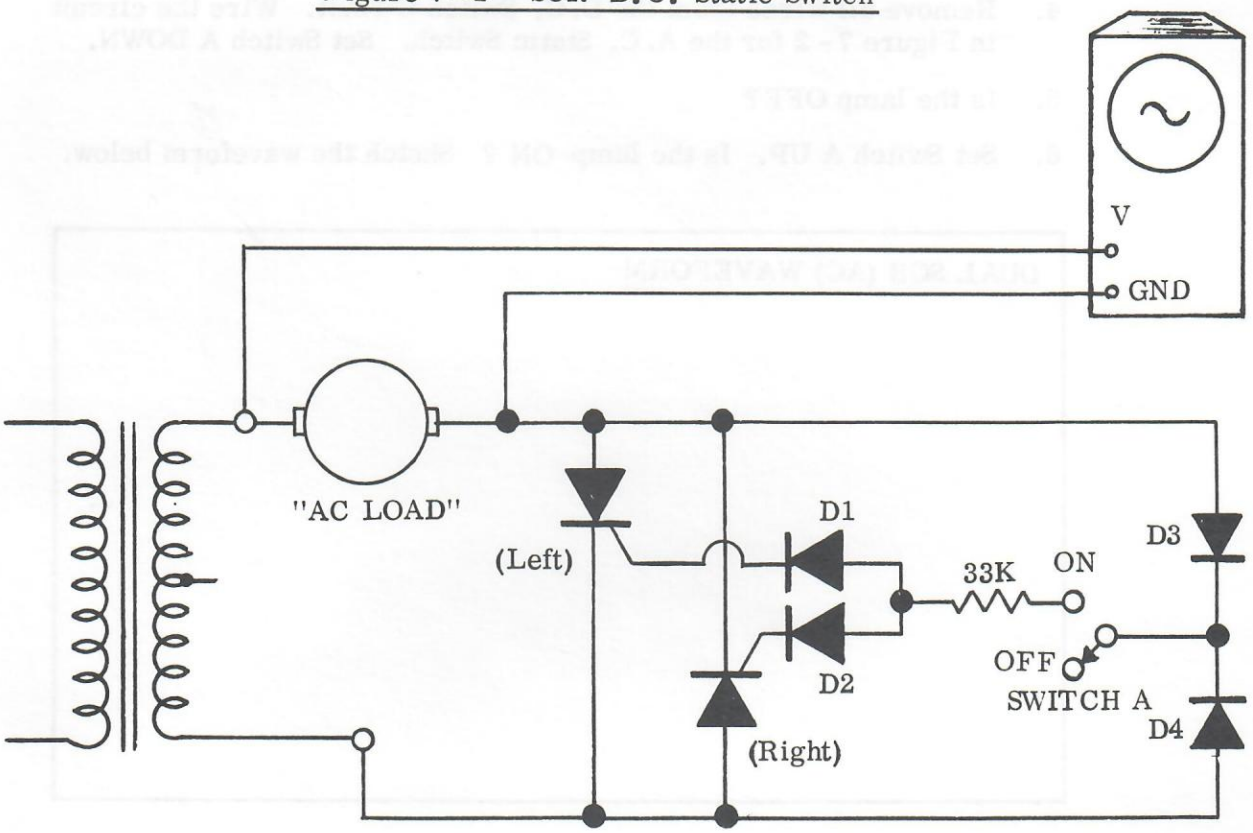



Figure 7 - 2 SCR A.C. Static Switch

Objective:

You will use an SCR as an ON-OFF switch with a low current gate input in a D.C. circuit. You will use two SCRs in an ON-OFF A.C. Circuit using a low current gate control.

Procedure:

1. Wire the circuit in Figure 7 - 1. Set Switches A DOWN and Switch B DOWN.
2. Set Switch A UP, then DOWN. A positive voltage on the gate will turn the SCR ON. Does the lamp turn on?
3. The Capacitor is charged to the full D.C. voltage through the 100K resistor. When Switch B is placed UP, the positive terminal of the capacitor is connected to ground resulting in a drop of the full D.C. volts at both ends of the capacitor including the SCR Anode. This shuts the SCR off into the "blocked" state.

   Set Switch B UP then DOWN. Does the LAMP turn OFF?
4. Remove all wires from the D.C. Switch Circuit. Wire the circuit in Figure 7 - 2 for the A.C. Static Switch. Set Switch A DOWN.
5. Is the lamp OFF?
6. Set Switch A UP. Is the lamp ON ? Sketch the waveform below.

DUAL SCR (AC) WAVEFORM

**Ed-Lab** | UNIJUNCTION TRANSISTOR | EXP. NO. 8 Page No. 1

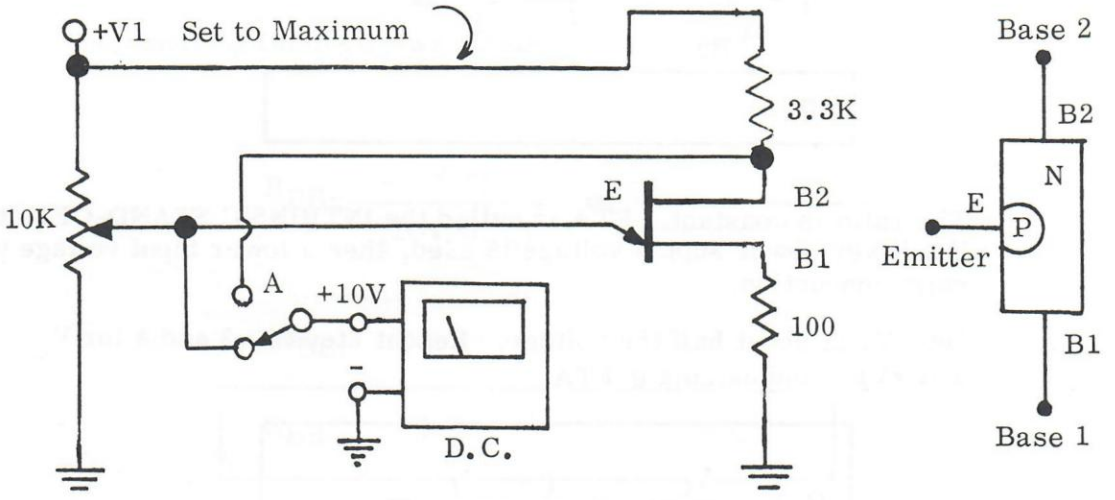



Figure 8-1 Unijunction Emitter Input

Objective:

You will adjust the input "emitter (E)" voltage to a Unijunction Transistor and observe its operation. The Emitter, E, is "open circuit" until a certain percentage voltage is reached at which time "E" is "connected" to B1.

Procedure:

1. Wire the circuit and set the 10K POT input voltage to zero. Set Switch A DOWN to monitor the Emitter voltage. Set +V1 to maximum.

2. Raise the input voltage using the 10K POT until the transistor conducts as detected when the voltage suddenly drops. Measure this voltage. (Measure voltage before it "drops".)

| $V_C$ = |
|---|

3. Lower the INPUT VOLTAGE (10K POT) to zero. Set Switch A UP and measure the Unijunction Supply Voltage at Base 2 (B2).

| $V_{B2}$ = |
|---|

4. Calculate the ratio ( $\eta$ is Greek letter ETA):

Pronounced: ("ate-uh").

$$\eta = \frac{V_C}{+V_{B2}} = \frac{(\quad)}{(\quad)} =$$

$\eta$ =

The ratio is constant. ETA is called the INTRINSIC STAND-OFF RATIO. If a lower power supply voltage is used, then a lower input voltage will start conduction.

5. Set +V1 at about half the voltage. Repeat steps 2, 3 and 4 for $V_C$ and $+V_{B2}$ and calculate ETA.

$$\eta = \frac{V_C}{V_{B2}} = \frac{(\quad)}{(\quad)} =$$

6. How does this value of ETA compare with that calculated in 4 above?

7. When the Unijunction Transistor is not conducting, there is an INTERBASE RESISTANCE, $R_{BB}$, between base B2 and B1. This resistance can be measured using a simple voltage divider. Set +V1 to maximum.
Set Switch A, UP. Set the 10K POT to zero. Read the output voltage at Base B2.

$V_{B2}$ =

8. Remove the +10V lead (from the meter) and from Switch A.

9. Using a separate lead wire, from +10V on the meter, measure the supply voltage.

+V1 =

10. The Base 2 Voltage is given by the voltage divider equation:

$$V_{B2} = +V1 \quad \frac{R_{BB}}{R_{BB} + 3.3K}$$

(neglecting the 100 ohm resistor)

Solving for $R_{BB}$, we obtain:

$$R_{BB} = \frac{3.3K\ V_{B2}}{(+V1 - V_{B2})} = \text{_____}$$

Solve for $R_{BB}$:

| $R_{BB}$ = |
|---|

| Ed-Lab | UNIJUNCTION SWEEP | EXP. NO. 9<br>Page No. 1 |
|---|---|---|

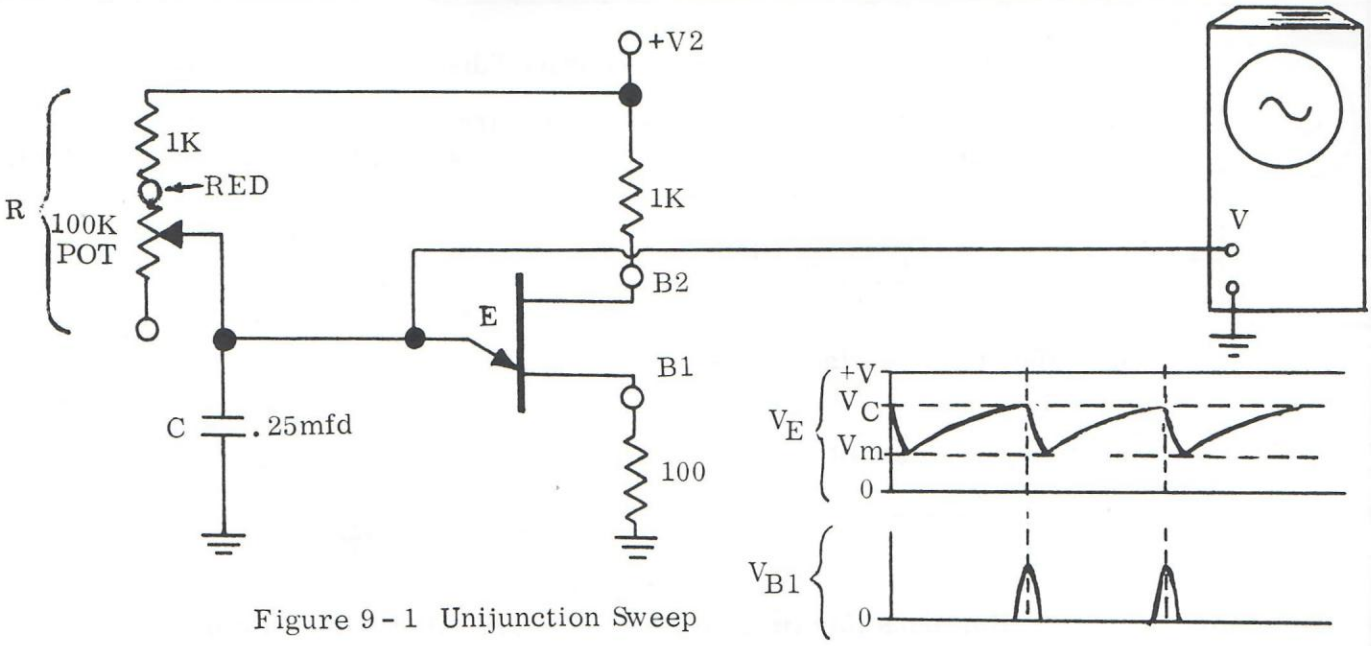



Figure 9 - 1 Unijunction Sweep

Objective:

You will observe the waveforms from a Unijunction Sweep Circuit where a capacitor is "charged" through the 100K POT and then "discharged" when the Emitter Voltage reaches the voltage where the emitter conducts. This operation continues and a "sweep" voltage is generated. This circuit is called a RELAXATION OSCILLATOR.

Procedure:

1. Wire the circuit in Figure 9 - 1. Set the 100K POT at 1/2 scale.
2. Observe and Plot the waveforms at points B2, E, and B1.

| WAVEFORM E |
|---|
| WAVEFORM B2 |
| WAVEFORM B1 |

3. Be sure to observe the short positive "discharge" pulse at point B1.

4. Set the POT at midway or 50K. Measure the Sweep Time or Period if a time base calibrated oscilloscope is available (Monitor waveform E).

| PERIOD = |
|---|

5. The time constant is $\tau = RC$

$$\tau = RC = 50KC \times .25 \text{ mfd}$$

$$= 50\ 10^{3} \times (.25)\ 10^{-6} \text{ seconds}$$

$$\tau = 12.5 \text{ milliseconds}$$

How does this time constant compare with the measured period?

6. While observing the waveform at point E reduce the 100K POT setting until the waveform stops sweeping (Rotate 100K POT clockwise.)

At this point, the series resistance is small enough to supply enough current to keep the emitter conducting and a voltage sweep will not occur. Estimate this minimum resistance setting:

| $R_{min}$ = |
|---|

7. Substitute a +10 MFD capacitor. What is the sweep period at a R = 100K setting?
Substitute a +100 MFD capacitor. What is the sweep period?

Add a +100 MFD cap in parallel (C=200MFD). What is the sweep period?

| Ed-Lab | UNIJUNCTION SCR CONTROL | EXP. NO. 10<br>Page No. 1 |
|---|---|---|

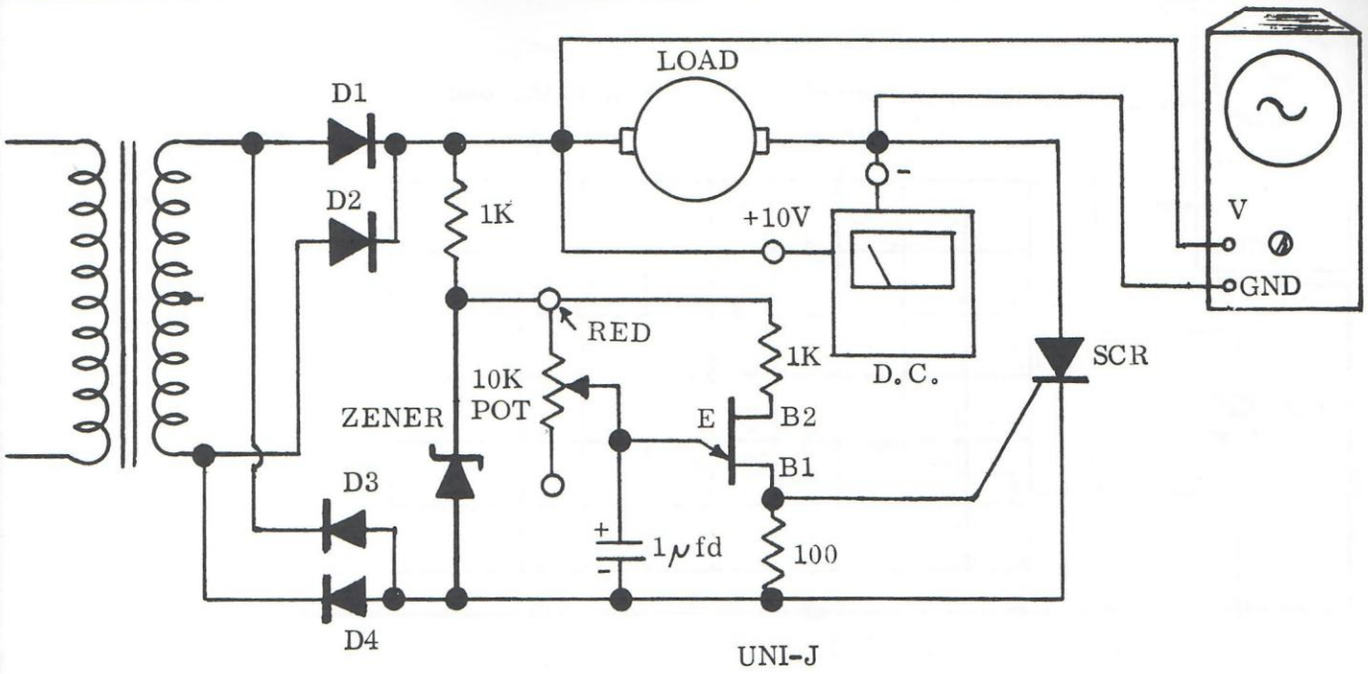



Figure 10 - 1 Unijunction SCR Control

Objective:

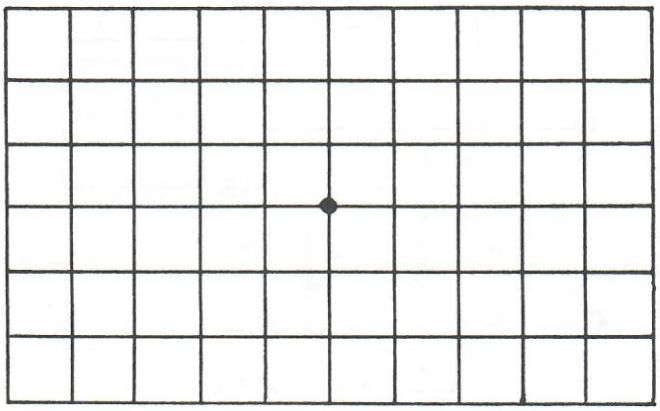
You will use a Unijunction Sweep to provide the Delay to trigger the SCR. This circuit provides a better control of the DELAY and of the LOAD voltage.

Procedure:

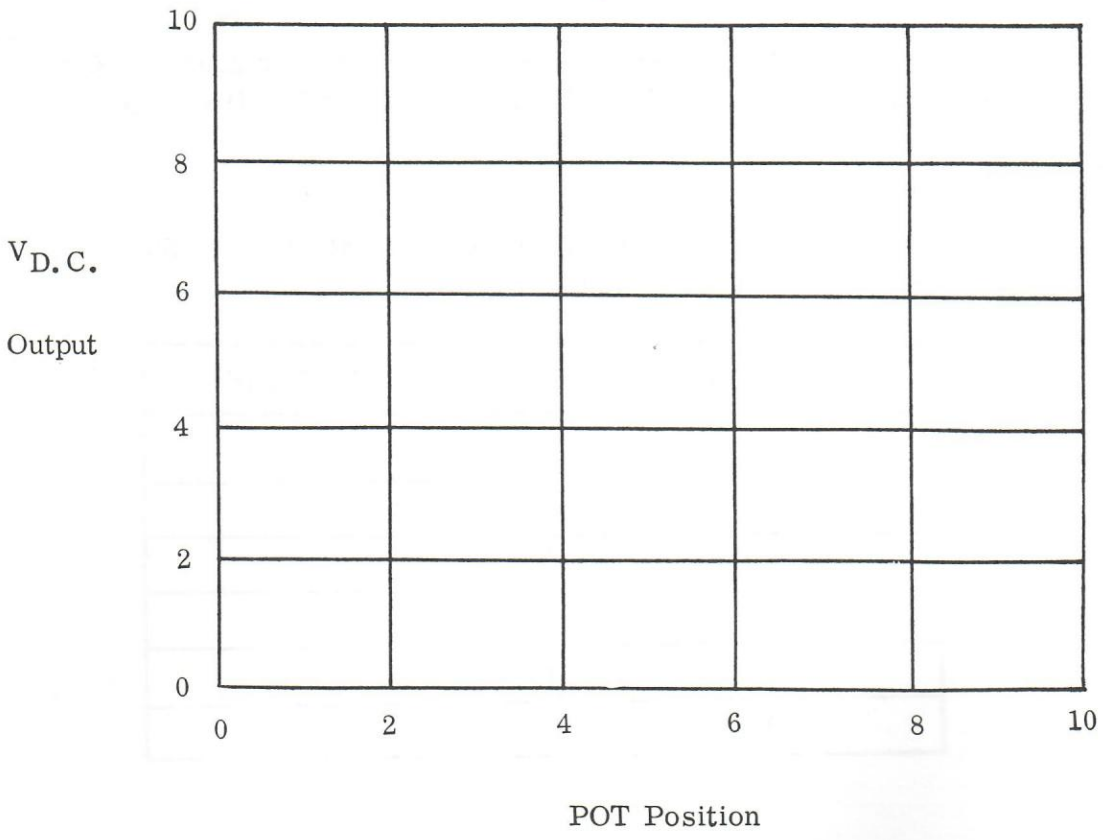
1. Wire the circuit in Figure 10 - 1.
2. Vary the 10K POT, monitor the waveshape and measure the D.C. Voltage. Enter the data below:

| POT POSITION | D.C. VOLTAGE |
|---|---|
| 0 | |
| 2 | |
| 4 | |
| 6 | |
| 8 | |
| 10 | |

3. Plot one of the waveforms as seen on the oscilloscope.



4. Plot the $V_{DC}$ outputs versus POT POSITION.



| Ed-Lab | UNIJUNCTION DELAY-ON CONTROL | EXP. NO. 11<br>Page No. 1 |
|---|---|---|

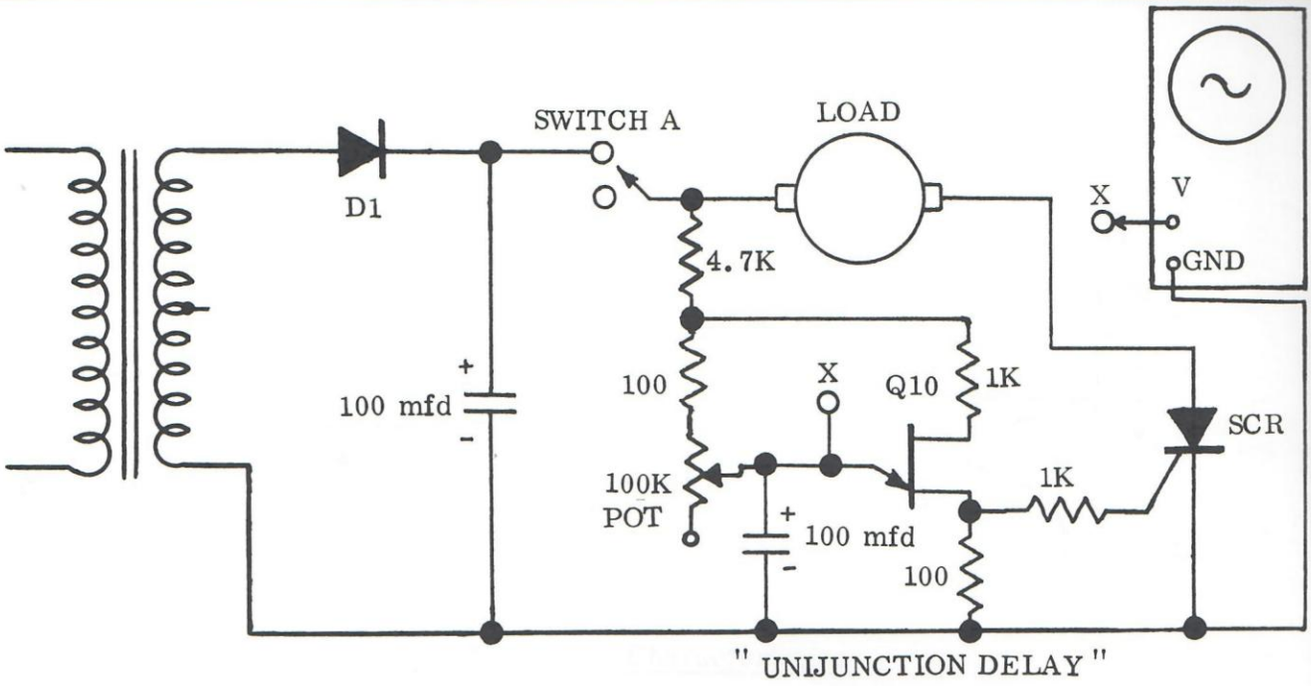



Figure 11-1 Delay Circuit

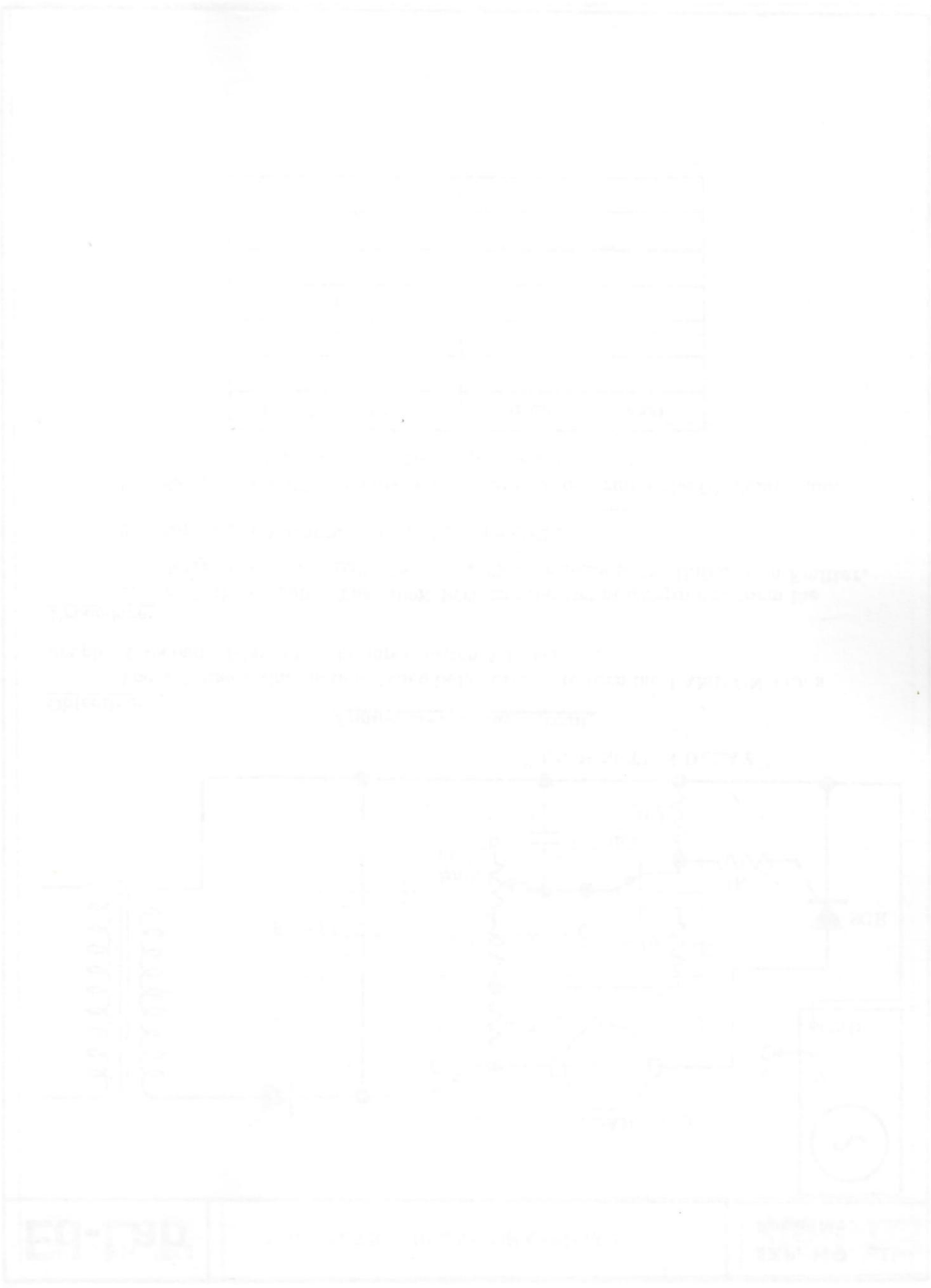
Objective:

You will use a Unijunction Sweep Delay circuit to turn the LAMP ON with a couple of seconds delay after the input Switch A is set UP.

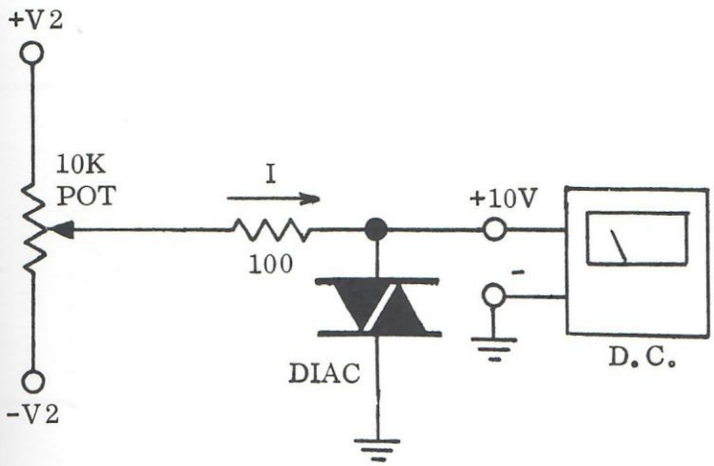
Procedure:

1. Wire the circuit. The 100K POT and the 100 mfd capacitor form the delay "time-constant". The Scope "X" connects to the Unijunction Emitter.
2. Set switch A DOWN to turn the lamp OFF.
3. Set switch A UP, and use a wristwatch to determine the ON delay time. Plot the delays below as the pot position is varied.

| POT POSITION | DELAY (seconds) |
|---|---|
| 0 | |
| 2 | |
| 4 | |
| 6 | |
| 8 | |
| 10 | |



| Ed-Lab | DIAC OPERATION | EXP. NO. 12<br>Page No. 1 |
|---|---|---|



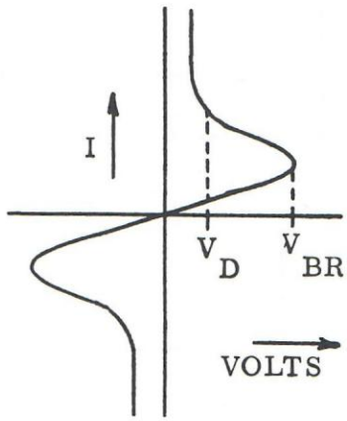




Figure 12-1 DIAC Characteristics

Objective:

You will observe the operation of the DIAC. As the DIAC voltage is increased, a voltage is reached where the diode "breaks-down" ($V_{BR}$), and then the voltage drops (to $V_D$).

Procedure:

1. Wire the circuit. The Meter measures the voltage across the resistor which is proportional to the DIAC Current.

2. Increase the voltage from 0 and measure the two diode voltages:

| |
|---|
| $V_{BR}$ = |
| $V_D$ = |

3. Reverse the meter leads and observe the same operation in the negative direction.

| Ed-Lab | DIAC CHARACTERISTIC CURVE | EXP. NO. 13<br>Page No. 1 |
|---|---|---|

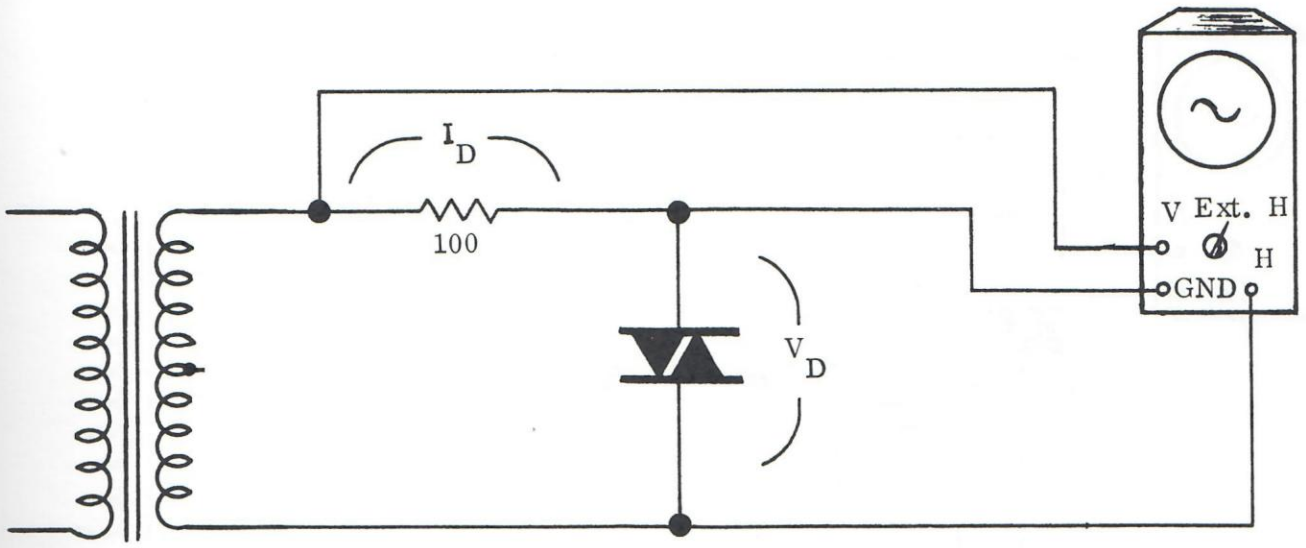


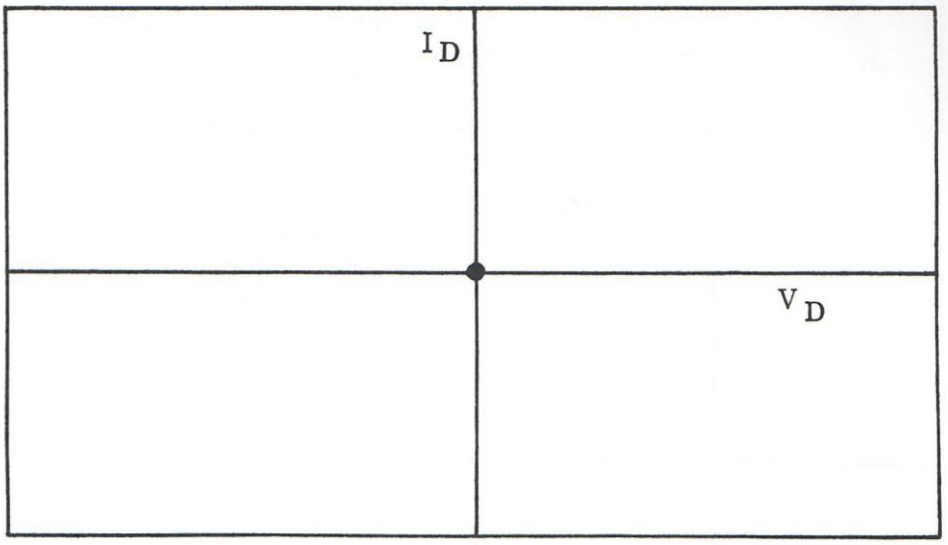

Figure 13-1 DIAC Characteristic Curve

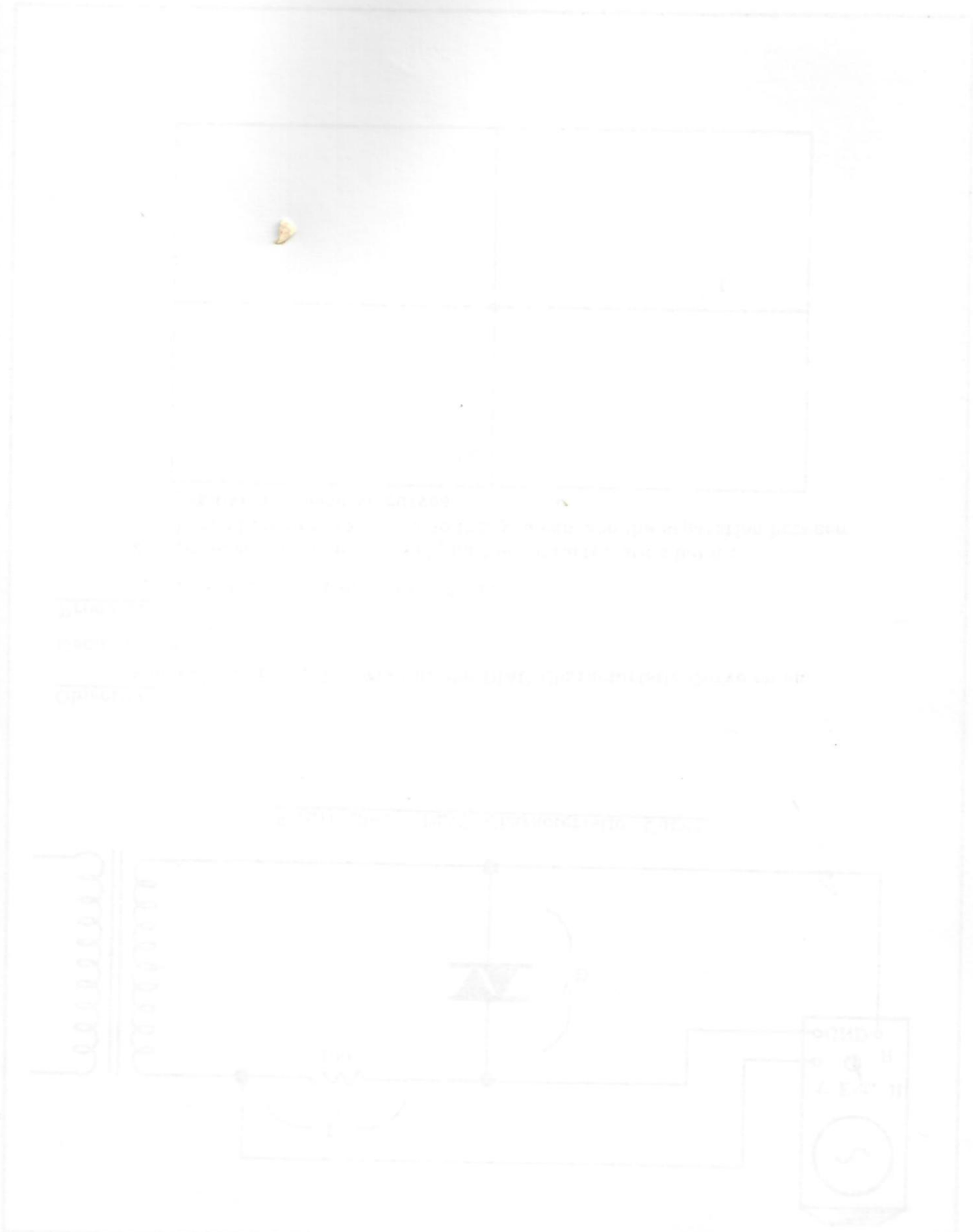
Objective:

You will automatically generate the DIAC Characteristic Curve on an Oscolloscope.

Procedure:

1. Wire the circuit in Figure 13-1.
2. Observe the waveform and plot the characteristics below:
   Expand the vertical scale so that you can see the separation between positive and negative curves.





| Ed-Lab | TRIAC OPERATION | EXP. NO. 14<br>Page No. 1 |
|---|---|---|

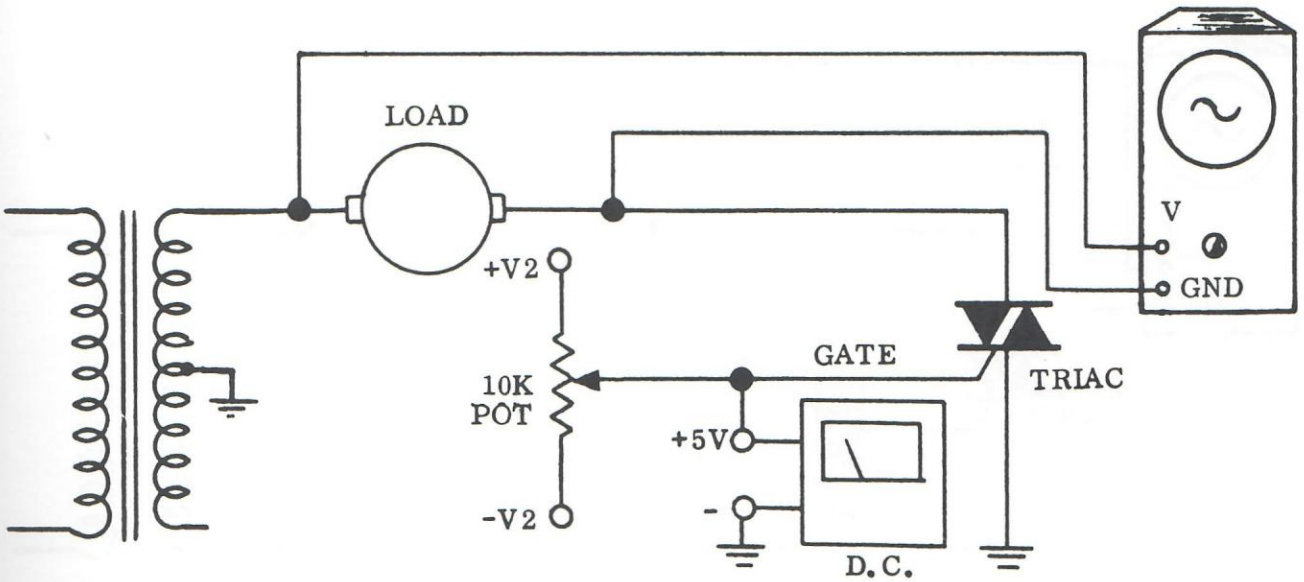



Figure 14-1 TRIAC Gate Control

Objective:

You will check the operation of the TRIAC. The TRIAC can be considered as a "two-way" SCR. You will apply a GATE voltage and observe that the TRIAC conducts in both directions, i.e., A.C. Current.

Procedure:

1. Wire the circuit. Use the 10K POT (in mid-position) to set the gate voltage at 0.
2. Increase the voltage and observe that the TRIAC is "closed (conducting)" for a positive or negative voltage for the positive and negative gate inputs.
3. Measure the "turn-on" voltage:

| | |
|---|---|
| $V_{T\ pos}$ = | |
| $V_{T\ neg}$ = | |

4. Plot the waveforms below:

| | | | | |
|---|---|---|---|---|
| POSITIVE GATE | | | | |
| NEGATIVE GATE | | | | |

| Ed-Lab | TRIAC PHASE SHIFT CONTROL | EXP. NO. 15<br>Page No. 1 |
|---|---|---|

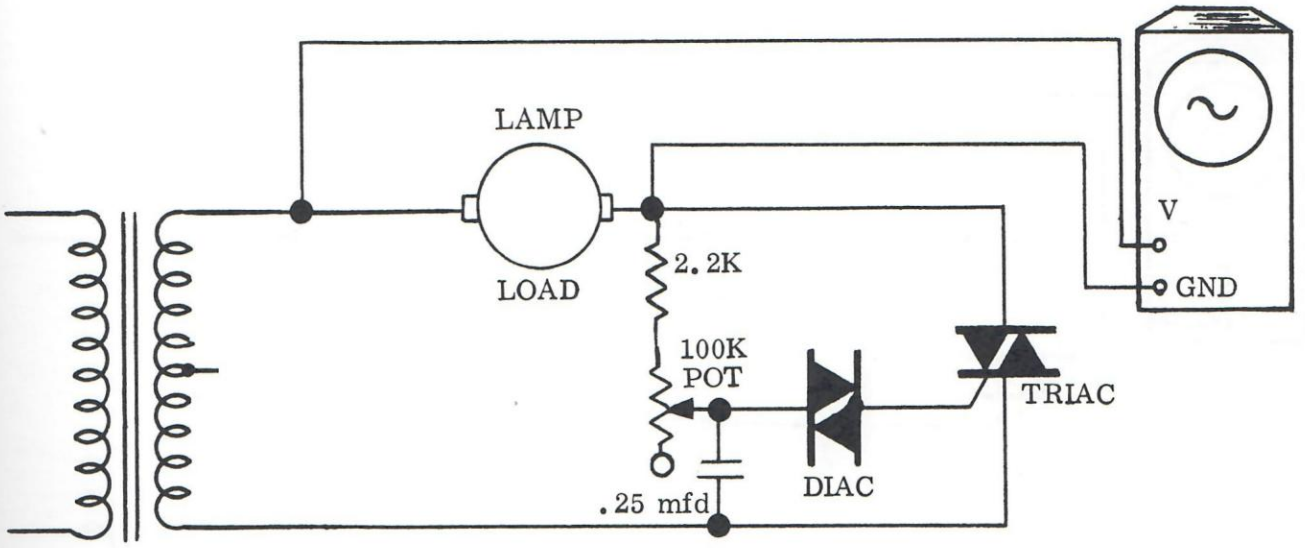



Figure 15 - 1 TRIAC - DIAC Phase Shift Circuit

Objective:

You will use a PHASE-SHIFT network and DIAC to trigger the TRIAC. The full transformer voltage is used because there is a minimum voltage required to trigger the DIAC. Using the full transformer voltage results in a greater variation in A.C. power.

Procedure:

1. Wire the circuit in Figure 15 - 1.

2. Vary the 100K POT and observe the output waveforms, plot waveforms below.

| Lamp | Waveform |
|---|---|
| DIM | |
| MEDIUM | |
| BRIGHT | |

| Ed-Lab | TRIAC LATCHING AND HOLDING CURRENTS | EXP. NO. 16<br>Page No. 1 |
|---|---|---|

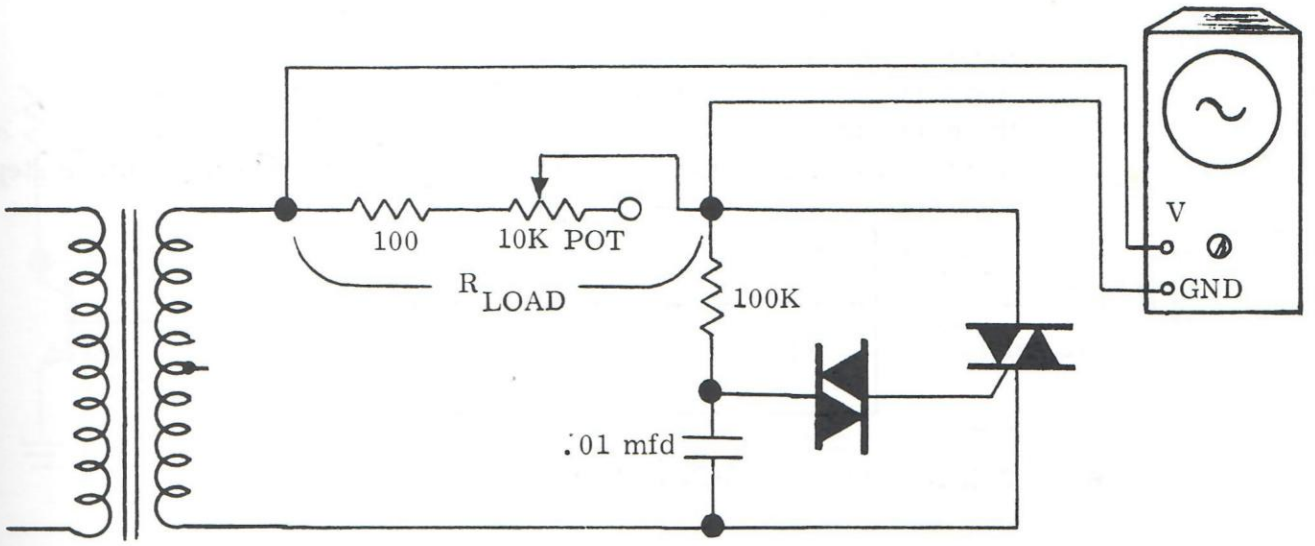



Figure 16 - 1

Objective:

You will decrease the TRIAC Load Current by increasing the load resistor and observe that there is a minimum HOLDING CURRENT required to keep the TRIAC ON and a minimum LATCHING CURRENT required to turn the TRIAC ON. The Latching Current is larger than the Holding Current.

Procedure:

1. Wire the above circuit which has a 100 ohm resistor in series with a 10K POT as a LOAD. Set the 10K POT to zero.
2. Observe the waveform with the 10K POT set to 0.
3. Increase the 10K POT resistor LOAD and observe the waveform. The decreasing sine-wave stops HOLDING at low voltage.

   Set the 10K POT at 2K (2 divisions). Measure the voltage just before the decreasing sine wave seen on the scope drops abruptly to zero.

   $V_H$ =

4. Calculate the HOLDING CURRENT.

$$I_H = \frac{V_H}{2K} = \frac{V_H}{2} \text{ (in milliamperes)}$$

$I_H$ = ma

5. Increase the 10K POT Load resistance further until the waveform just turns ON. The waveform (TRIAC) should be ON. This gives the minimum LATCHING CURRENT. Read the POT setting $R_L$ and the Latching Voltage level which is the initial positive voltage step.

| |
|---|
| $R_L$ = |
| $V_L$ = |

6. Calculate the LATCHING CURRENT

$$I_L = \frac{V_L}{R_L} = \frac{(\quad)}{(\quad)}$$

| |
|---|
| $I_L$ = ma |

**Ed-Lab** | DIGITAL ASTABLE MULTIVIBRATOR | **EXP. NO.** 17 **Page No.** 1

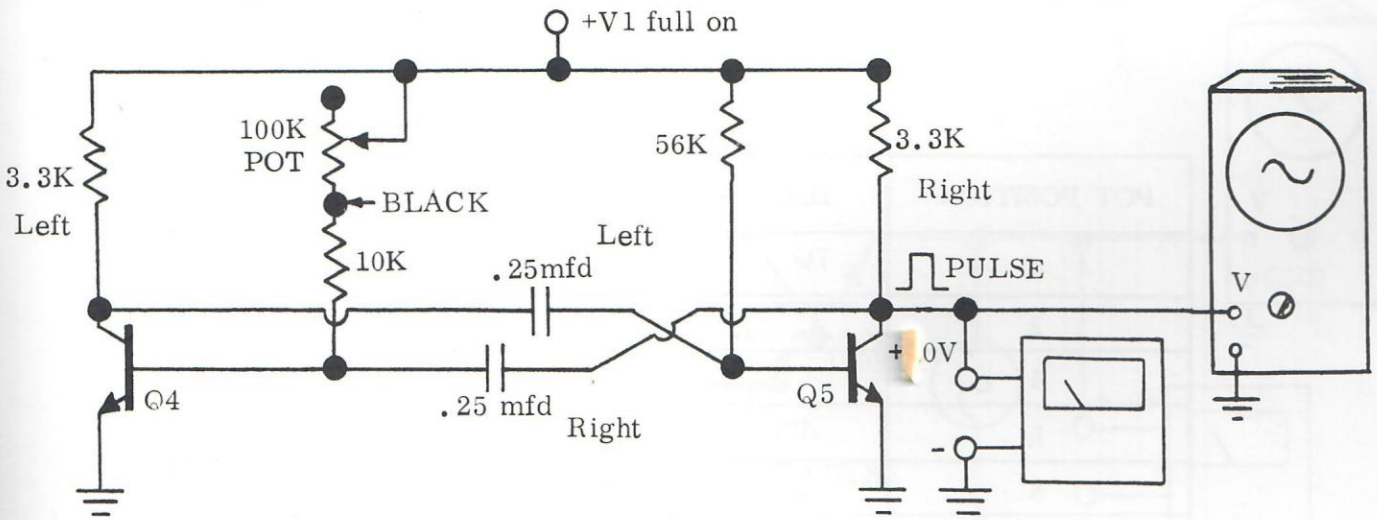



Figure 17 - 1 Astable Multivibrator

Objective:

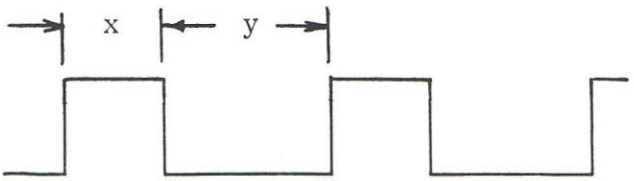
You will use two transistors and two R-C delay networks in an Astable Multivibrator circuit, where the Voltage alternates between plus voltage and zero voltage.

Procedure:

1. Wire the circuit in Figure 17 - 1.

2. View the output on the Oscilloscope.

   Vary the 100K POT and observe the waveform:




   The ratio of the plus time "x" to the total time (x+y) is called the "duty cycle". For the waveform shown, the duty cycle is

$$\text{DUTY CYCLE} = \frac{x}{x+y} \times 100\% = \frac{1}{3} \times 100\%$$

$$\text{DUTY CYCLE} = 33\%$$

3. The Duty Cycle is proportional to the D.C. Voltage reading. Vary the 100K POT and observe the waveform and "estimate" the duty cycle. Read the D.C. voltage and enter it in the table on the following page:

| POT POSITION | D.C. VOLTAGE | Estimate of Duty Cycle |
|---|---|---|
| 0 | | |
| 2 | | |
| 4 | | |
| 6 | | |
| 8 | | |
| 10 | | |

**Ed-Lab** | DIGITAL SERIES VOLTAGE CONTROL | **EXP. NO.** 18 **Page No.** 1

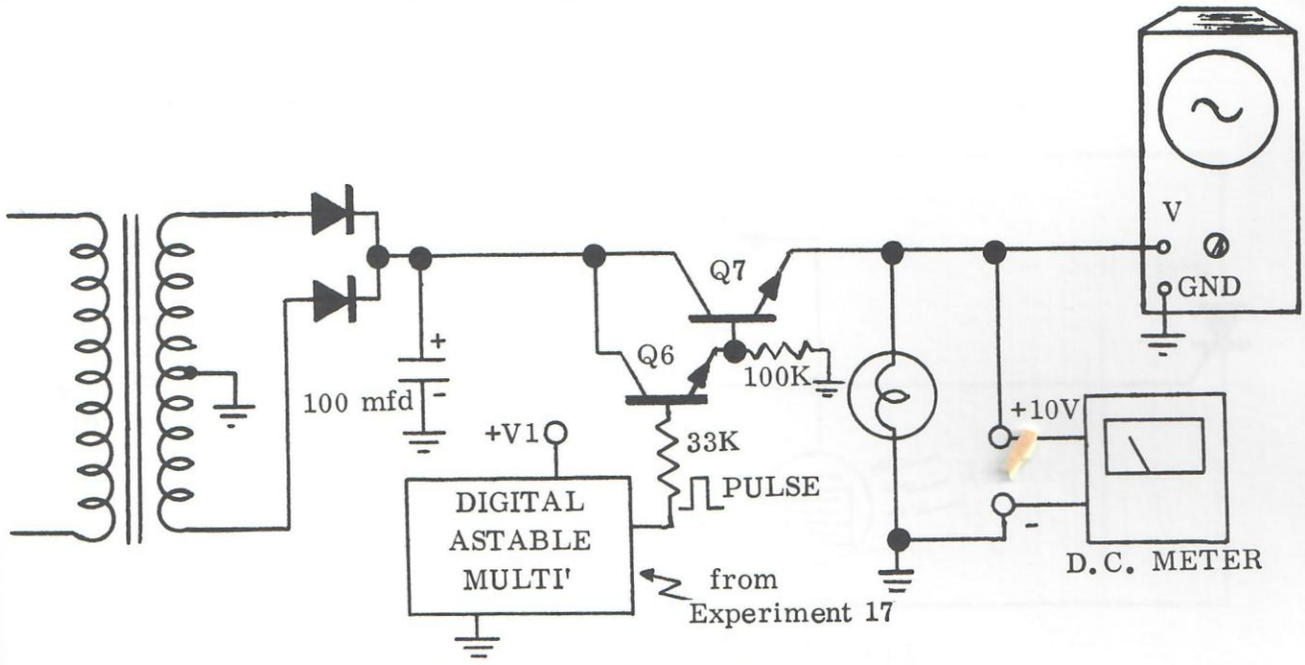



Figure 18-1 Digital Voltage Control

Objective:

A series transistor is used to control the output voltage. The series transistor uses a "Darlington Driver" for a low base current drive. The transistor is either FULL-ON or OFF. When it is OFF, there is no current and no lost power. When it is FULL-ON, there is a small voltage drop and therefore only a small power dissipation or loss. The Digital Supply thus has a very high efficiency and uses a low-power transistor to provide high current. The Astable controls the on-off cycle.

Procedure:

1. Wire the circuit in Figure 18 - 1.
2. Vary the Control POT and observe the lamp waveform and the lamp D.C. Voltage. Enter the data below:

| CONTROL POT 100K (ASTABLE) POT | LAMP Brightness | D.C. VOLTAGE |
|---|---|---|
| 0 | | |
| 2 | | |
| 4 | | |
| 6 | | |
| 8 | | |
| 10 | | |

| Ed-Lab | PHOTOCELL SCR CONTROL | EXP. NO. 19<br>Page No. 1 |
|---|---|---|

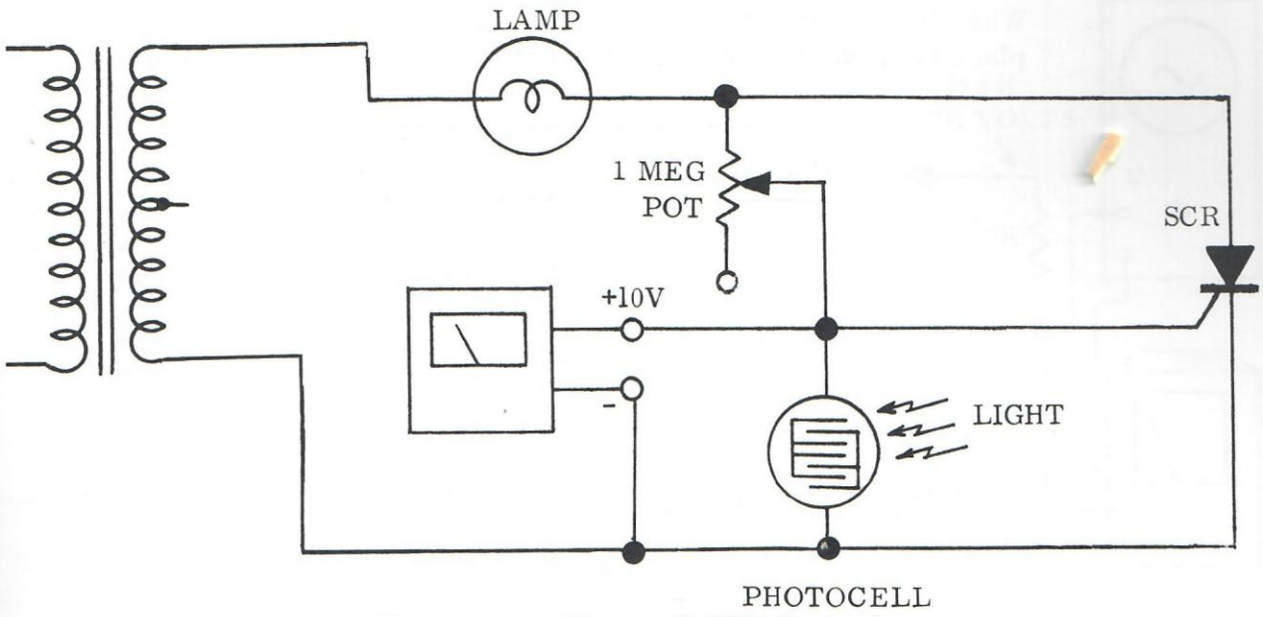



Figure 19-1 Photocell SCR Control

Objective:

You will use a PHOTOCELL to detect a change in LIGHT and activate the SCR to turn the LAMP ON.

Procedure:

1. Wire the circuit in Figure 19 - 1.
2. Adjust the 1 M POT until the lamp just turns OFF. Keep away from the photocell.
3. Place your hand near the Photocell. The Photocell resistance is increased and the gate voltage is increased to turn-on the SCR. Check the operation:

| | | | Check |
|---|---|---|---|
| Step 1 | ADJUST 1 MEG POT | LAMP OFF | |
| Step 2 | HAND OVER Photocell | LAMP ON | |

4. What is the furthest distance away from the photocell that you can place your hand to decrease the light enough to turn the lamp ON?

| DISTANCE = |
|---|

**Ed-Lab** | STEP-UP TRANSFORMER | **EXP. NO.** 20 **Page No.** 1

* EXPERIMENTS 20 THRU 29 REQUIRE THE USE OF AN EXTERNAL TRANSFORMER MODULE.

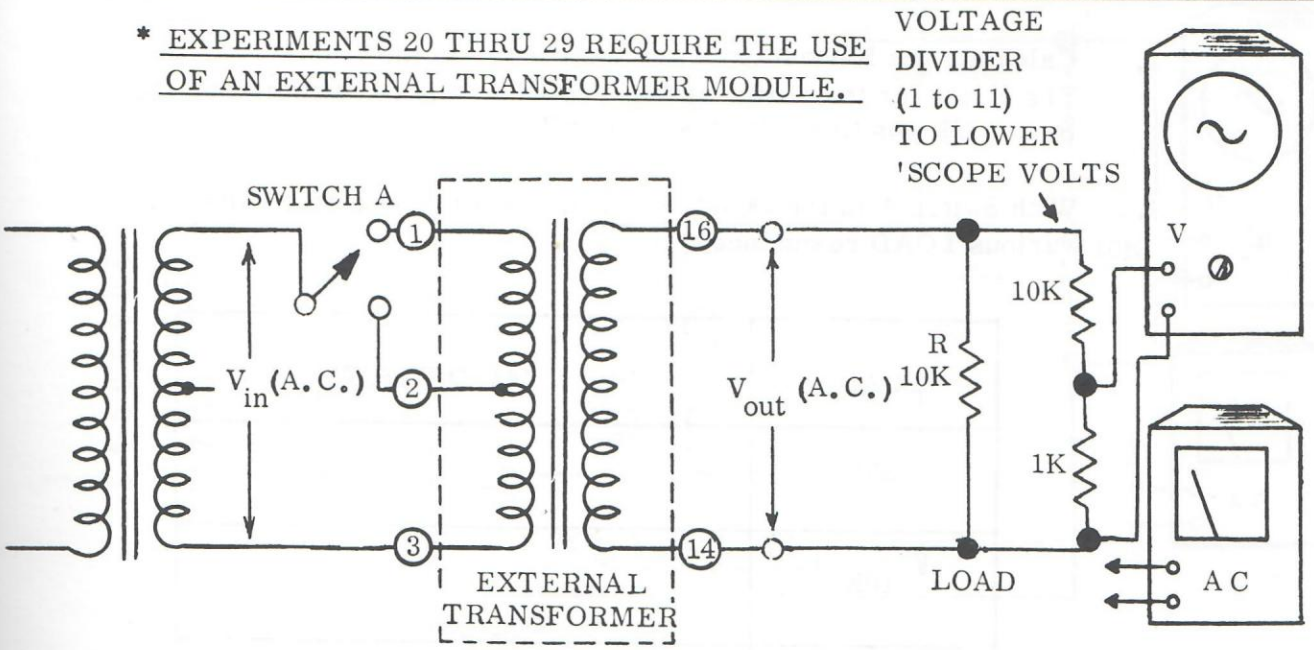



Figure 20 - 1 Step-up Transformer

Objective:

You will apply an A.C. Input Voltage to a TRANSFORMER and measure the A.C. Output Voltage at the secondary winding.

Procedure:

WARNING: THE TRANSFORMER PRODUCES HIGH VOLTAGES. DO NOT TOUCH LEADS WITH YOUR FINGERS. TURN THE TRAINER OFF WHEN CHANGING WIRES.

1. Wire the circuit. The TURNS RATIO can be changed using Switch A by applying the input across the full winding (UP) or across the half winding (DOWN).
2. Measure the input and output voltages as indicated in the table and enter the results below: Use scope or AC meter for AC voltages.

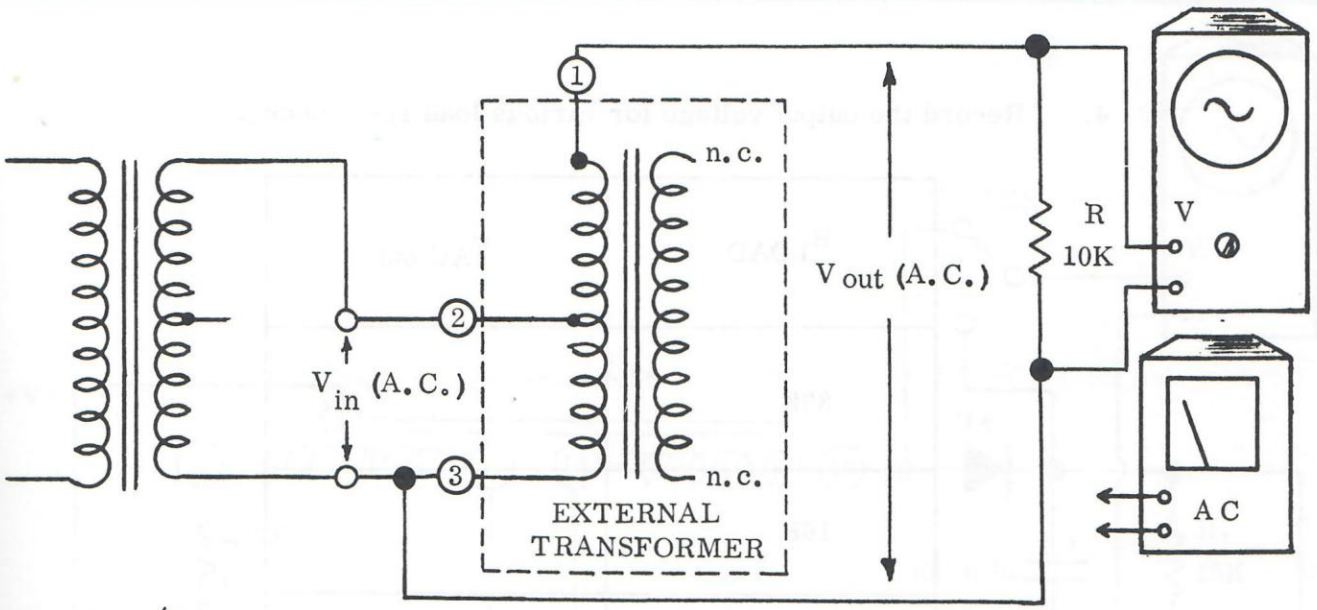
| set | measure | | calculate |
|---|---|---|---|
| Switch A | $V_{input\ A.C.}$ | $V_{output\ A.C.}$ (times 11) | Ratio = $\frac{V_{output\ A.C.}}{V_{input\ A.C.}}$ |
| UP | | | |
| DOWN | | | |

3. Calculate the Ratios and enter the results in the table.
The Ratio for the Switch DOWN should be higher than that for the Switch UP position. Is this correct?

4. With Switch A in the DOWN position, record the output voltage for various LOAD resistances.

| $R_{LOAD}$ | $V_{AC\ OUTPUT}$ |
|---|---|
| 33K | |
| 10K | |
| 3.3K | |
| 1K | |

**Ed-Lab** | AUTO-TRANSFORMER | **EXP. NO.** 21 **Page No.** 1




n.c./no connection

Figure 21 - 1 Auto - Transformer

Objective:

You will use only one "winding" on the transformer with the input at the "center tap" and achieve a voltage gain across the full winding.

Procedure:

1. Wire the circuit. Use the External Transformer. Observe that only one winding is used.

2. Measure the input and output A.C. Voltages. Enter the data below:

| |
|---|
| $V_{in}$ A.C. = |
| $V_{out}$ A.C. = |

3. Calculate the Voltage Gain:

$$\text{GAIN} = \frac{V_{out}\ \text{A.C.}}{V_{in}\ \text{A.C.}} =$$

Because the center tap is in the middle of the winding, the output has twice the windings compared to the input and the theoretical Gain should be 2. How does the measured Gain compare to this theoretical value?

4. Record the output voltage for various load resistances:

| $R_{LOAD}$ | $V_{AC\ out}$ |
|---|---|
| 33K | |
| 10K | |
| 3.3K | |
| 1K | |

| Ed-Lab | RINGING - CHOKE CONVERTER/INVERTER | EXP. NO. 22<br>Page No. 1 |
|---|---|---|

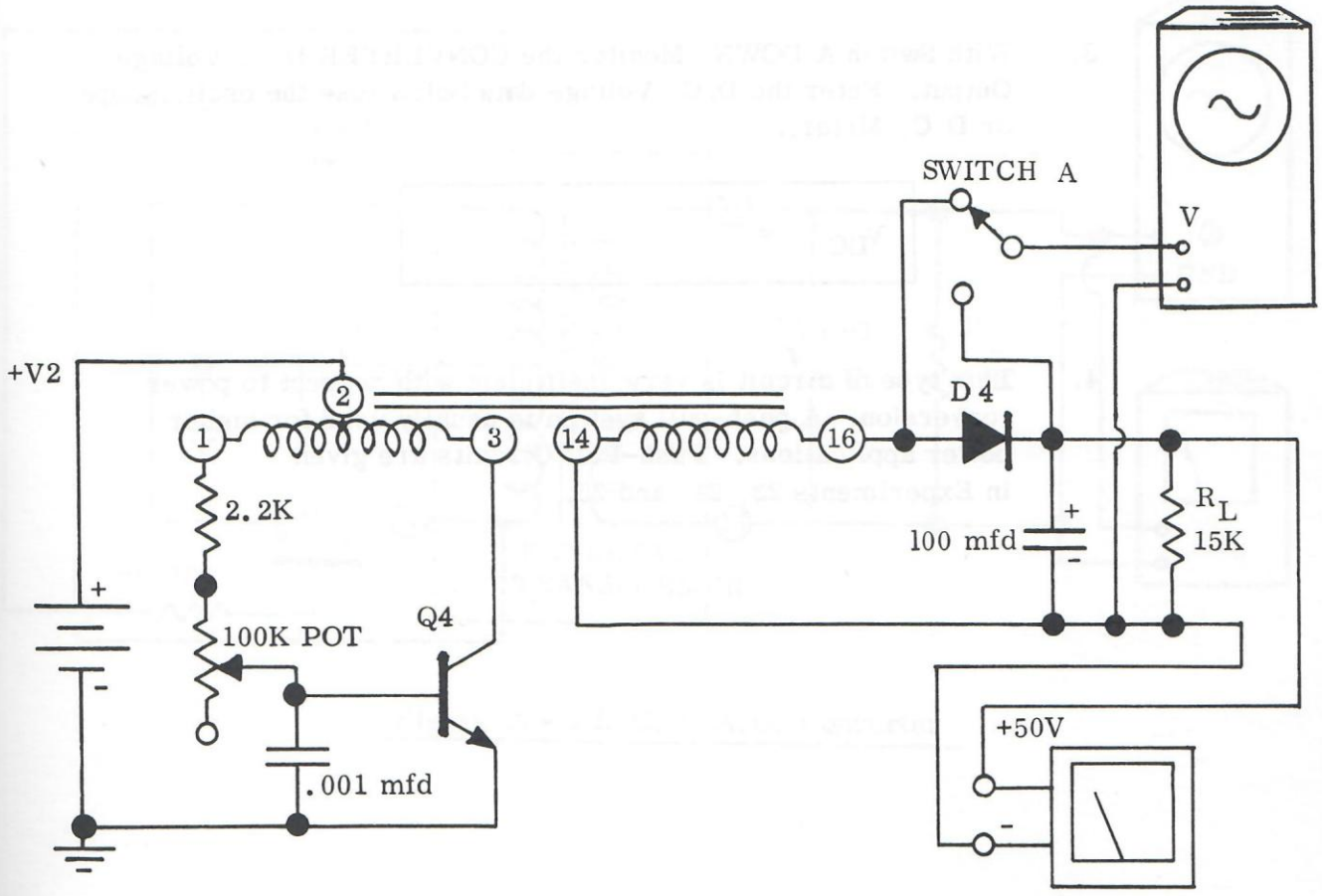



Figure 22-1 Oscillator Converter

Objective:

You will use a Transistor and Transformer Oscillator Circuit to change the D.C. Voltage Supply to an A.C. VOLTAGE (INVERTER) and to a different D.C. VOLTAGE (CONVERTER).

Procedure:

1. Wire the circuit in Figure 22-1. This circuit is a Feedback Oscillator with a transformer coupled output.

2. With Switch A UP, use the oscilloscope to monitor the output A.C. Voltage. Read the INVERTER Voltage output and enter this data below:

| $V_{AC}$ = |
|---|

3. With Switch A DOWN, Monitor the CONVERTER D.C. Voltage Output. Enter the D.C. Voltage data below (use the oscilloscope or D.C. Meter):

| $V_{DC}$ = |
|---|

4. This type of circuit is very inefficient with respect to power conversion. A push-pull system is usually used for higher power applications. Push-Pull Circuits are given in Experiments 23, 24, and 25.

| Ed-Lab | D.C. TO A.C. CONVERTER | EXP. NO. 23<br>Page No. 1 |
|---|---|---|

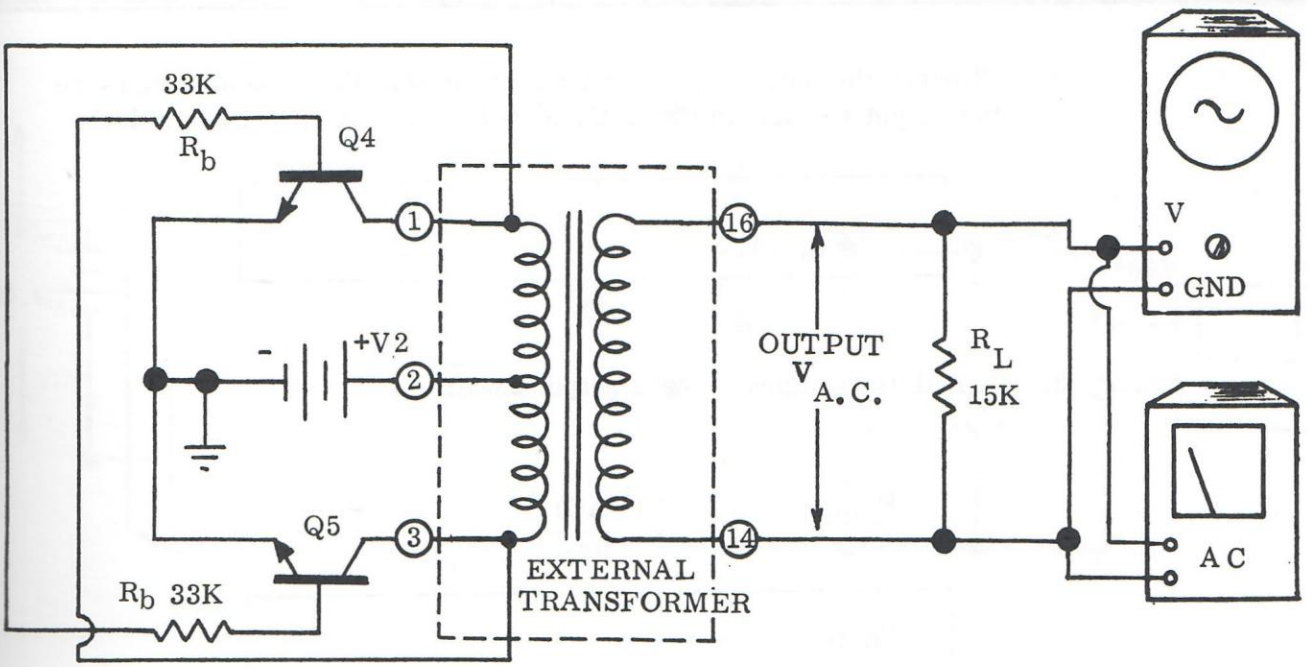



Figure 23 - 1 D.C. to A.C. Converter

Objective:

You will use a TRANSFORMER and TRANSISTORS as an OSCILLATOR to convert a D.C. Voltage (or BATTERY VOLTAGE) to an A.C. Voltage. Typical applications would be to convert a 12 Volt Automobile Battery output to 115 Volts, 60 Hertz A.C. This Converter is also referred to as an INVERTER because it "inverts" D.C. to A.C.

Procedure:

1. Wire the above circuit. This Converter uses a single transofrmer. Other converters have more windings, and others use two transformers. The frequency of oscillation in this circuit is determined by the Inductance (L) of the Transformer and the Resistor (R) as well as the effective load resistance of the transformer. Other converter circuits use capacitors where the frequency is determined by the capacitor and resistor. The values of R and L used above correspond to a frequency of approximately 60 cylces per second for the load used.

2. Observe the output A.C. Voltage on the Oscilloscope and measure the output voltage on the A.C. Meter. Enter this voltage below:

| Output $V_{A.C.}$ = |
|---|

3. Calculate the approximate output power

$$P_{output} = \frac{(V_{AC})^2}{R_L}$$

| $P_{output}$ = |
|---|

4. Substitute other load resistances and observe the variation in output Voltage.

| $R_L$ | Output $V_{A.C.}$ |
|---|---|
| 100K | |
| 56K | |
| 15K | |
| 10K | |
| 1K | |

5. Substitute other base resistors $R_b$ such as 10 K , 3.3K .

Comment on the results.

| Ed-Lab | D.C. TO D.C. CONVERTER | EXP. NO. 24<br>Page No. 1 |
|---|---|---|

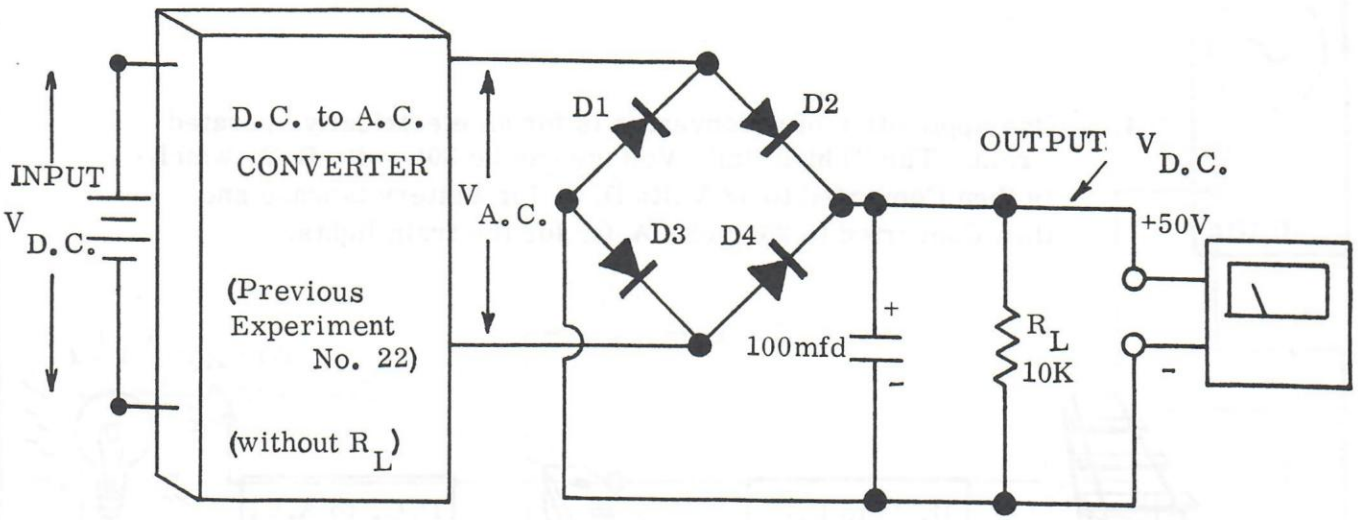



Figure 24 - 1 D.C. to D.C. Converter

Objective:

You will add a BRIDGE RECTIFIER to the D.C. to A.C. Converter of the previous experiment to convert D.C. from one Voltage to another D.C. Voltage.

Procedure:

1. Wire the above circuit.
2. Measure the input and output D.C. Voltages

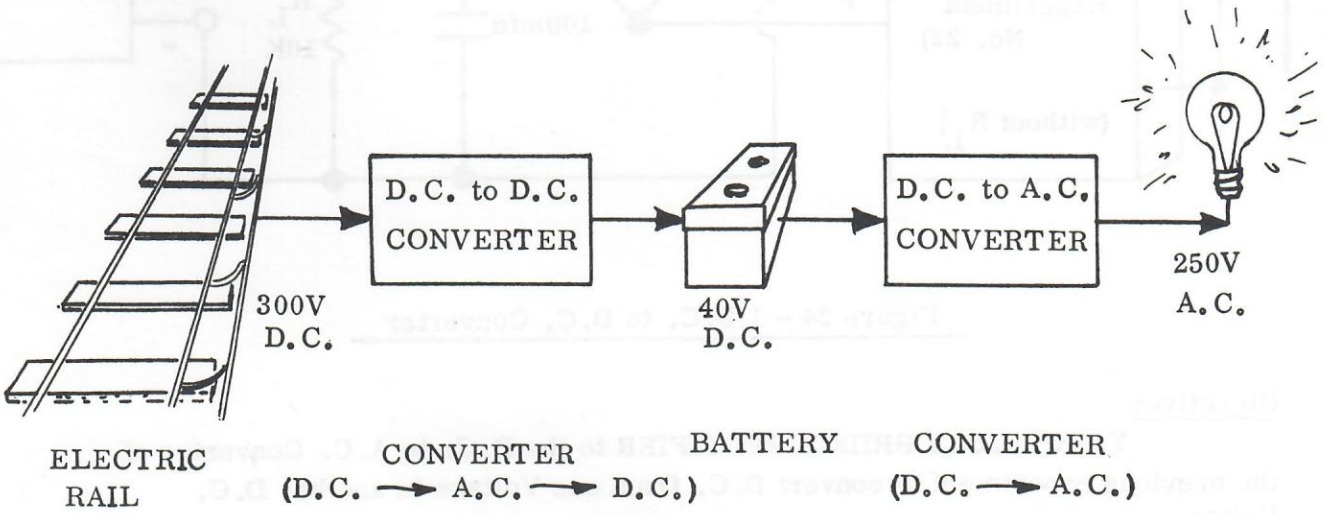
| | | |
|---|---|---|
| Input | $V_{D.C.}$ | = |
| Output | $V_{D.C.}$ | = |

3. Calculate the ratio of the output to input voltages:

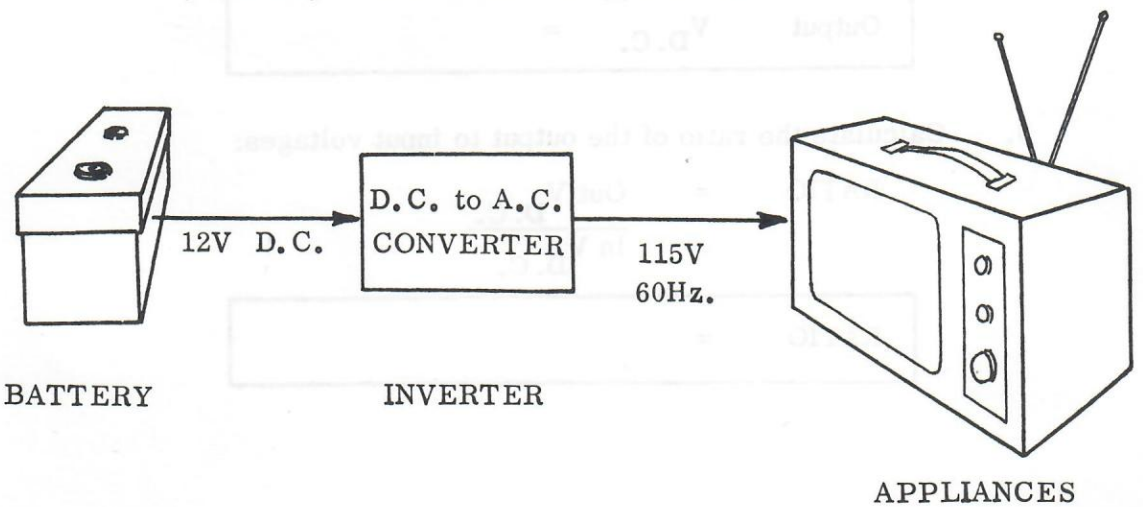
$$\text{RATIO} = \frac{\text{Out } V_{D.C.}}{\text{In } V_{D.C.}}$$

| | | |
|---|---|---|
| RATIO | = | |

4. One application of a Converter is for an electrically operated Train. The "Third Rail" Voltage can be 300 volts D.C. which is then Converted to 48 Volts D.C. for Battery Storage and then Converted to 250 volts A.C. for the train lights.




5. Another application is for Mobile Conversion System between a 12 Volt Battery in an Automobile or in a Boat to 115 Volts to permit operation of normal appliances.



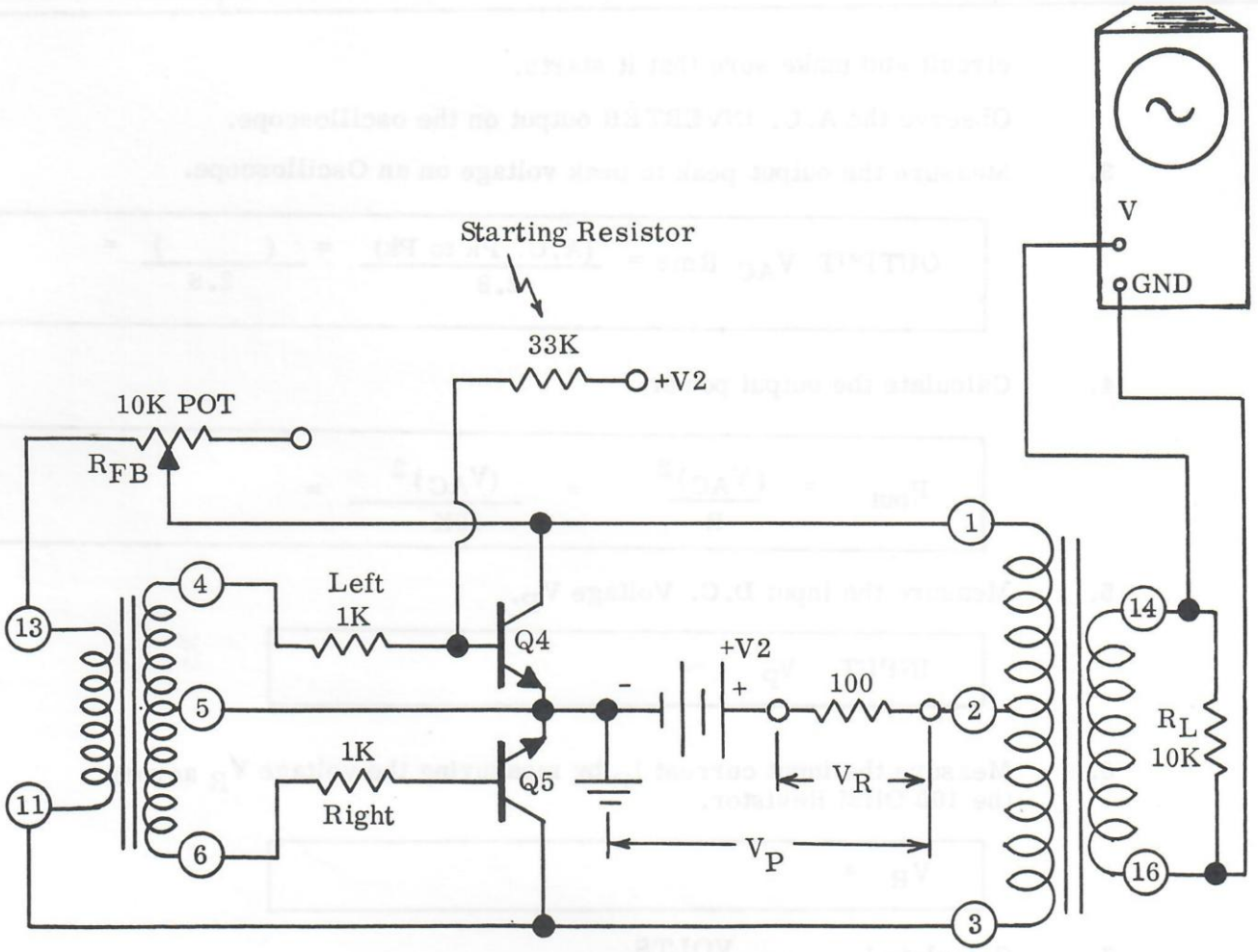



Figure 25-1 Two-Transformer Inverter

**Objective:**

You will test a Two-Transformer Push-Pull Inverter. Two transformers provide a separate high power output transformer, and a low power input transformer for improved power conversion efficiency.

**Procedure:**

1. Wire the circuit in Figure 25-1.
2. The feedback 10K POT (RFB) is in series with the input transformer to reduce the collector-to-collector voltage to the proper voltage required for the transformer. The setting of the 10K POT will also affect the frequency. Set the 10K POT halfway. The circuit may normally not start. The 33K resistor is inserted to unbalance the

circuit and make sure that it starts.

Observe the A.C. INVERTER output on the oscilloscope.

3. Measure the output peak to peak voltage on an Oscilloscope.

| |
|---|
| OUTPUT $V_{AC}$ Rms = $\frac{\text{(A.C. Pk to Pk)}}{2.8}$ = $\frac{(\quad)}{2.8}$ = |

4. Calculate the output power:

| |
|---|
| $P_{out} = \frac{(V_{AC})^2}{R} = \frac{(V_{AC})^2}{10K} =$ |

5. Measure the input D.C. Voltage $V_P$.

| |
|---|
| INPUT $V_P$ = |

6. Measure the input current $I_P$ by measuring the voltage $V_R$ across the 100 OHM Resistor.

| |
|---|
| $V_R$ = |

7. Calculate $I_P$.

$$I_P = \frac{V_R \text{ (VOLTS)}}{100} = V_R \times 10 \text{ ma} = (\quad) 10 \text{ ma}$$

| |
|---|
| $I_P$ = ma. |

8. Calculate the input power.

$$P_{in} = V_P I_P$$

| |
|---|
| $P_{in}$ = |

9. Calculate the Power Conversion Efficiency:

$$E = \frac{P_{out}}{P_{in}} \times 100\%$$

| |
|---|
| E = |

**Ed-Lab** | TRANSFORMER INDUCTANCE | **EXP. NO.** 26 **Page No.** 1

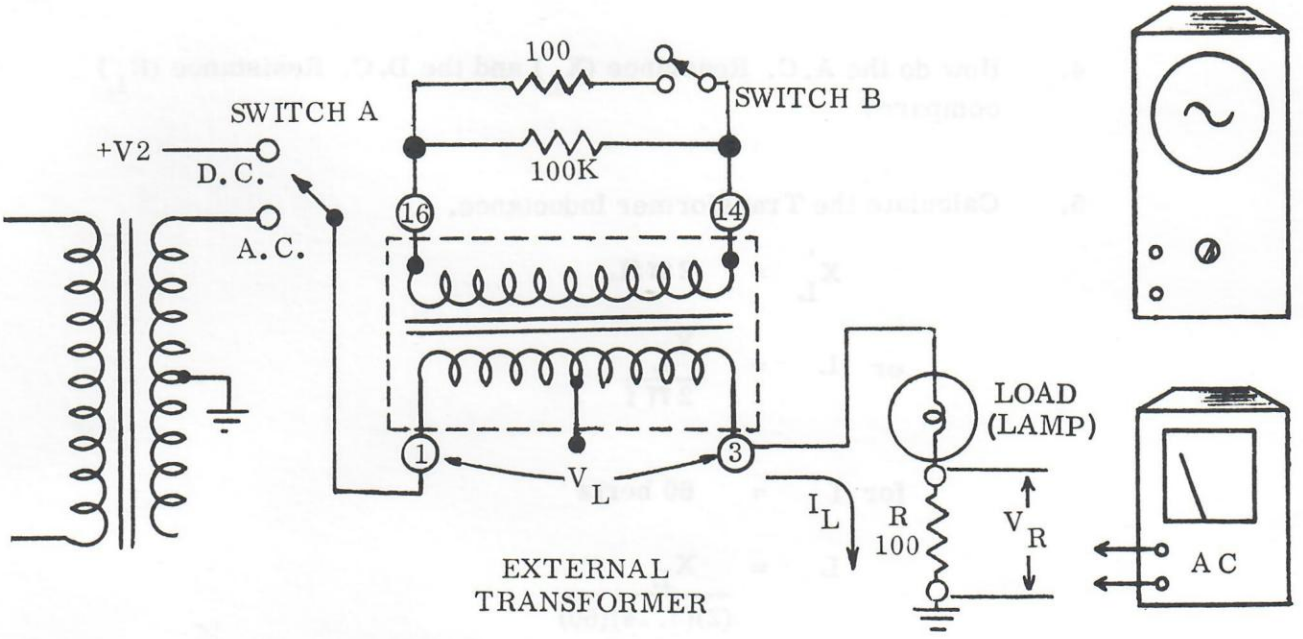



Figure 26-1 Inductance Measurements

Objective:

You will measure the Inductance (or A.C. Reactance) and the D.C. Resistance of the Transformer by placing the Transformer in a Series Circuit.

Procedure:

1. Wire the circuit. With Switch A UP, the input is a D.C. Voltage. With Switch A DOWN, the input is the A.C. Voltage. Keep Switch B UP with a load of 10K on the "coupled" winding.

2. Measure the A.C. and D.C. Voltages. Enter the data below:

| set | | measure | | calculate | | |
|---|---|---|---|---|---|---|
| Switch | Voltage | $V_R$ | $V_L$ | $I_L = \frac{V_R}{R} = \frac{V_R}{100}$ | $\frac{V_L}{I_L}$ | |
| Switch A UP | D.C. | | | | $R_L =$ | D.C. |
| Switch A DOWN | A.C. | | | | $X_L =$ | A.C. |

3. Calculate $R_L$ and $X_L$ in the above table.

| Ed-Lab | TRANSFORMER INDUCTANCE | EXP. NO. 26<br>Page No. 2 |
|---|---|---|

4. How do the A.C. Reactance ($X_L$) and the D.C. Resistance ($R_L$) compare?

5. Calculate the Transformer Inductance.

$$X_L = 2\pi fL$$

$$\text{or } L = \frac{X_L}{2\pi f}$$

$$\text{for } f = 60 \text{ hertz}$$

$$L = \frac{X_L}{(2)(3.14)(60)}$$

$$L = 2.66\ X_L \text{ millihenries}$$

where $X_L$ is in OHMs (Use $X_L$ from the previous page).

| L = |
|---|

The inductance results from the coil windings around the iron core of the transformer. By varying the Current in the winding 16-14, the "saturation" of the iron core changes and the device is called a Saturable Reactor.

6. Set Switch B DOWN. This applies a 100 Ω load to the transformer. This load allows current to flow in the 16-14 Winding, saturating the iron core and lowering the Inductance of the 1-3 Winding. Repeat the measurements of the Transformer and Resistor Voltages to determine Inductance. Enter the data below:

Switch A DOWN (A.C. Input)

Switch B DOWN (Load)

| measure | | calculate | | |
|---|---|---|---|---|
| $V_R$ | $V_L$ | $I_L = \frac{V_R}{R} = V_R \times 10$ ma | $X_L = \frac{V_L}{I_L}$ | mhy<br>$L_S = 2.66\ X_L$<br>SATURATED |
| | | | | $L_S =$ |

How does the Inductance with the Saturated Core calculated above compare with the Inductance without the saturated core as calculated in step 5? By varying the amount of saturation in a Saturable Reactor, the output Voltage is varied and a "Magnetic Amplifier" is devised. This is demonstrated in the next experiment. Summarize the measured Inductances below:

| | |
|---|---|
| As calculated in paragraph 5. | TRANSFORMER L = |
| As calculated in paragraph 6. | SATURATED L = |

**Ed-Lab** | A.C. MAGNETIC AMPLIFIER | **EXP. NO. 27** **Page No. 1**

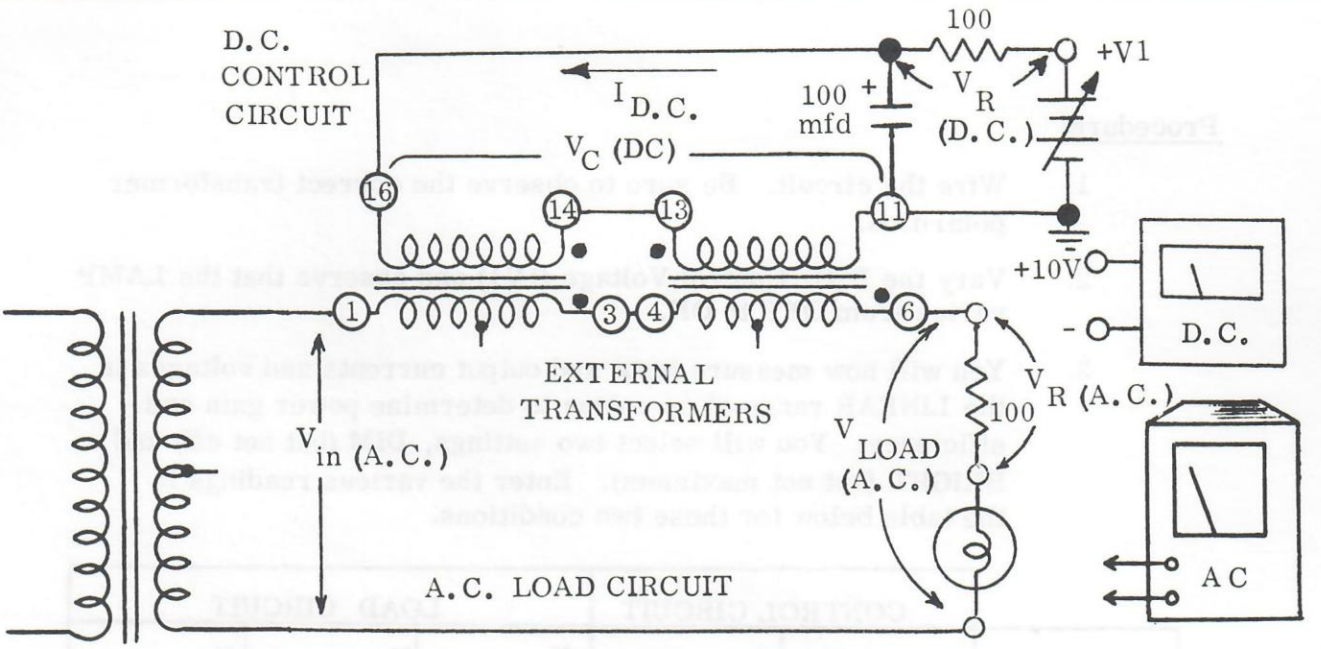



Figure 27-1 A.C. Magnetic Amplifier

Objective:

You will use two transformers or "saturable reactors" in a Magnetic Amplifier circuit to control A. C. Power. You will measure the Power Amplification and Efficiency.

Description:

Two transformers are connected in series (LOW TURNS WINDING - center taped) with the A.C. source (LOW VOLTAGE FROM THE ED-LAB TRANSFORMER) and in series with the LOAD (Lamp and 100 Ohms). The 100 ohm resistor enables you to measure the load current. The transformer "Impedance" is HIGH and thus there is little A.C. current flow to the Load. The secondary windings (high turns) are connected in SERIES in OPPOSING directions (see dots) so that the transformer induced A.C. voltages (currents) are cancelled in the CONTROL circuit. The D.C. Voltage +V1 is variable producing a variable D.C. Current and thus D.C. Saturation in the transformer Core. As in the previous experiment, current in the high-turns winding reduces the Series Inductance and will therefore result in a higher voltage and power in the output. Thus, a D.C. Current in the Control Circuit, is "amplified" to higher A.C. Current and Power in the Load. In practical Magnetic Amplifiers, standard transformers are not used, but special transformers have all windings on a single core and special core saturation characteristics are used.

Procedure:

1. Wire the circuit. Be sure to observe the correct transformer polarities.

2. Vary the D.C. Control Voltage (+V1) and observe that the LAMP varies from OFF to ON.

3. You will now measure input and output currents and voltages in the LINEAR range of operation to determine power gain and efficiency. You will select two settings, DIM (but not off) and BRIGHT (but not maximum). Enter the various readings in the table below for these two conditions.

| | CONTROL CIRCUIT | | LOAD CIRCUIT | | |
|---|---|---|---|---|---|
| Load | $V_{C(D.C.)}$ | $V_{R\ (D.C.)}$ | $V_{in\ (A.C.)}$ | $V_{Load(A.C)}$ | $V_{R\ (A.C.)}$ |
| DIM | | | | | |
| BRIGHT | | | | | |

The current is $\frac{V}{R}$ and for the control circuit:

$$I_{D.C.} = \frac{V_{R\ D.C.}}{100} = \frac{V_{R.D.C.}}{100} \times 1000 \text{ ma} = V_{R\ D.C.} \times 10 \text{ ma.}$$

(Example: for $V_{R\ D.C.} = 3V$) $\quad I_{D.C.} = 3 \times 10 \text{ ma}$

$$I_{D.C.} = 30 \text{ ma.}$$

4. The A.C. Circuit Current is calculated in the same manner.

The Power is (I•V) and for I in milliamperes, Power is in milliwatts.

Perform the calculations as shown in the table on the following page:

| | LOAD | DIM (1) | BRIGHT (2) |
|---|---|---|---|
| CONTROL CIRCUIT (D.C.) | $I_{D.C.} = V_{R\ D.C.}$ x 10 ma | = | = |
| | $P_{D.C.} = I_{D.C.} \times V_{C\ (D.C.)}$ | $P_{D.C.\ 1}$ = | $P_{D.C.\ 2}$ = |
| LOAD CIRCUIT (A.C.) | $I_{A.C.} = V_{R(A.C.)}$ x 10 ma | = | = |
| | $P_{LOAD} = V_{LOAD\ A.C.} \times I_{A.C.}$ | $P_{LOAD\ 1}$ = | $P_{LOAD\ 2}$ = |
| | $P_{in} = V_{in\ A.C.} \times I_{A.C.}$ | $P_{in\ 1}$ = | $P_{in\ 2}$ = |

5. Power Gain:

The absolute Power Gain is the ratio of Power Output to Power Input. Calculate the Power Gain P.G. for both conditions:

$$P.G._1 = \frac{P_{Load\ 1}}{P_{D.C.\ 1}} =$$

$$P.G._2 = \frac{P_{Load\ 2}}{P_{D.C.\ 2}} =$$

How does this Power Gain for a Magnetic Amplifier compare to a Transistor or SCR circuit Power Gain? Why is Power Gain important?

The Efficiency (E) is the ratio of total output power to total input power.

$$E = \frac{P_{Load\ (A.C.)}}{P_{in\ (A.C.)} + P_{D.C.}} \times 100\%$$

Calculate the Efficiency in both cases:

$$E_1 = \frac{P_{Load\ 1} \times 100\%}{P_{in\ 1} + P_{D.C.1}} =$$

$$E_2 = \frac{P_{Load\ 2} \times 100\%}{P_{in\ 2} + P_{D.C.\ 2}} =$$

| |
|---|
| $E_1 =$ |
| $E_2 =$ |

Why is Efficiency important?

Although many new designs use Solid-State Controls for Power Amplifications, much existing equipment uses magnetic amplifiers. The basic requirement for Power Amplifiers is to proportionately control large amounts of electrical power from low level power sources, with high EFFICIENCY or with minimum of power that is not wasted power. This experiment operates on low power levels for safety and ease in measurements, but the general principles and performance factors apply to high power circuits.

**Ed-Lab** | D.C. MAGNETIC AMPLIFIER | **EXP. NO.** 28 **Page No.** 1

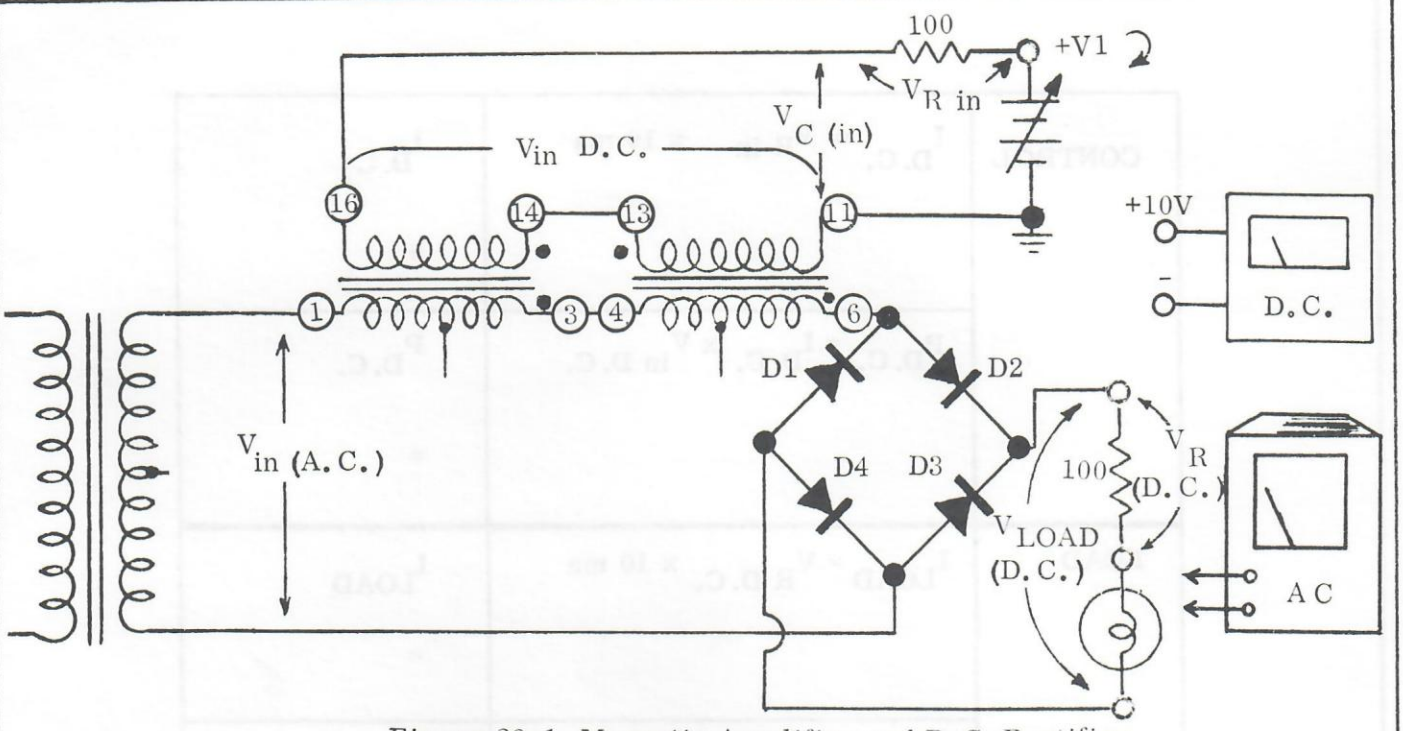



Figure 28-1 Magnetic Amplifier and D.C. Rectifier

Objective:

You will use a Magnetic Amplifier and Bridge Rectifier to provide D.C. Control of power. Efficiency and Power Gain are measured.

Procedure:

1. Wire the circuit and adjust the lamp for a medium brightness. Vary +V1.
2. Measure the voltages shown and enter the results in the table below:

| | Control Input | | Load Circuit | | |
|---|---|---|---|---|---|
| Load | $V_{in}$ D.C. | $V_{R\ in}$ | $V_{in}$ A.C. | $V_R$ D.C. | $V_{LOAD}$ D.C. |
| MED. BRIGHT | | | | | |

3. Calculate input and output power. (next page)

| | | |
|---|---|---|
| CONTROL | $I_{D.C.} = V_{R\ in}$ x 10 ma | $I_{D.C}$ = |
| | $P_{D.C.} = I_{D.C.} \times V_{in\ D.C.}$ | $P_{D.C.}$ = |
| LOAD | $I_{LOAD} = V_{R\ D.C.}$ x 10 ma | $I_{LOAD}$ = |
| | $P_{LOAD} = V_{LOAD} \times I_{LOAD}$ | $P_{LOAD}$ = |
| | $P_{INPUT} = V_{in\ A.C.} \times I_{Load}$ | $P_{INPUT\ A.C.}$ = |

4. Calculate the Power Gain and Efficiency:

$$\text{Power Gain} = \frac{P_{Load}}{P_{D.C.}\ \text{Control}} =$$

$$\text{Efficiency} = \frac{P_{Load}}{P_{in} + P_{CONTROL}} =$$

| Ed-Lab | FULL WAVE SCR BRIDGE | EXP. NO. 29<br>Page No. 1 |
|---|---|---|

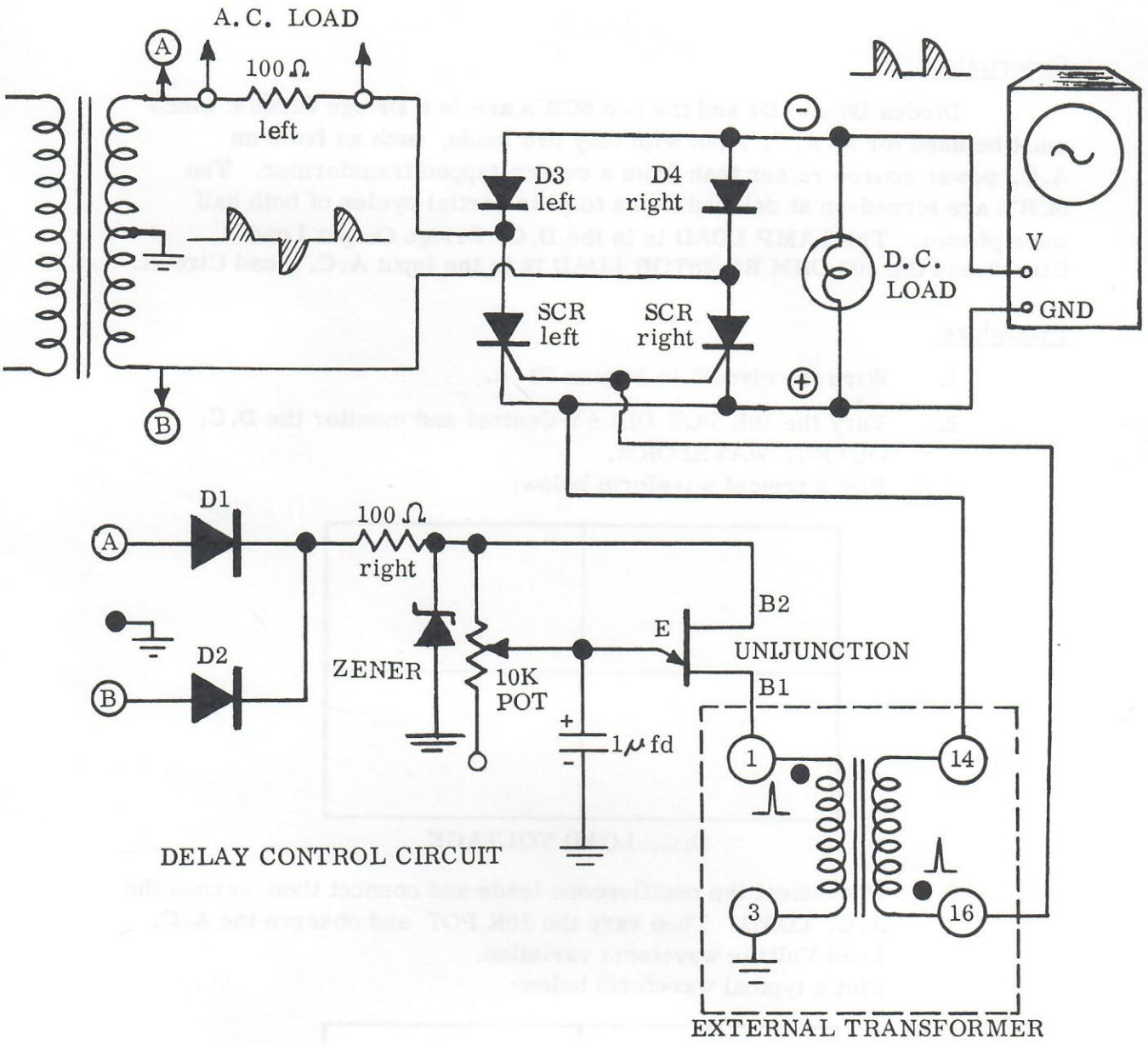



Figure 29 - 1 Full Wave SCR Bridge.

Objective:

You will use two SCR's in a FULL WAVE BRIDGE Circuit to provide a D.C. and an A.C. voltage Control using a UNIJUNCTION delay trigger with TRANSFORMER coupling.

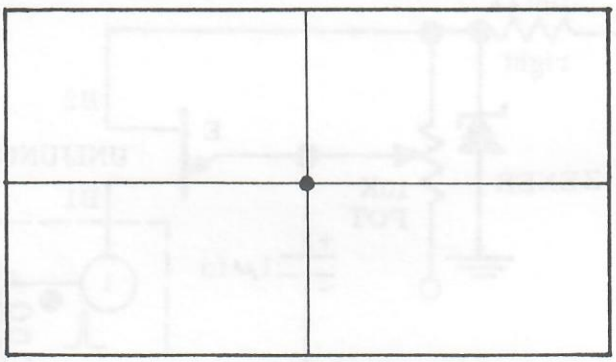
| Ed-Lab | FULL WAVE SCR BRIDGE | EXP. NO. 29<br>Page No. 2 |
|---|---|---|

Description:

Diodes D3 and D4 and the two SCR's are in a Bridge Circuit which must be used for an A.C. input with only two leads, such as from an A.C. power source rather than from a center-tapped transformer. The SCR's are turned-on at delayed times to pass partial cycles of both half wave phases. The LAMP LOAD is in the D.C. Bridge Output Load Circuit and the 100 OHM RESISTOR LOAD is in the Input A.C. Load Circuit.

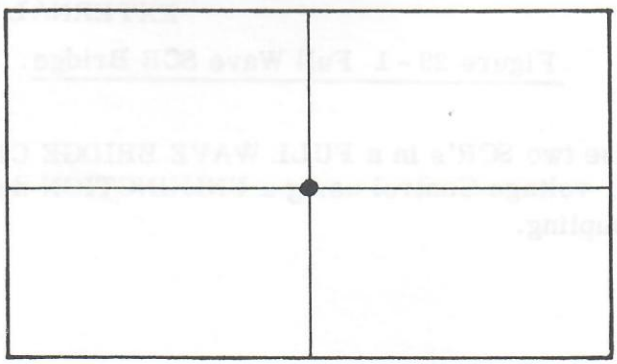
Procedure:

1. Wire the circuit in Figure 29 - 1.
2. Vary the 10K POT DELAY Control and monitor the D.C. OUTPUT WAVEFORM.
   Plot a typical waveform below:



D.C. LOAD VOLTAGE

3. Disconnect the oscilloscope leads and connect them across the A.C. LOAD. Then vary the 10K POT and observe the A.C. Load Voltage waveform variation.
   Plot a typical waveform below:

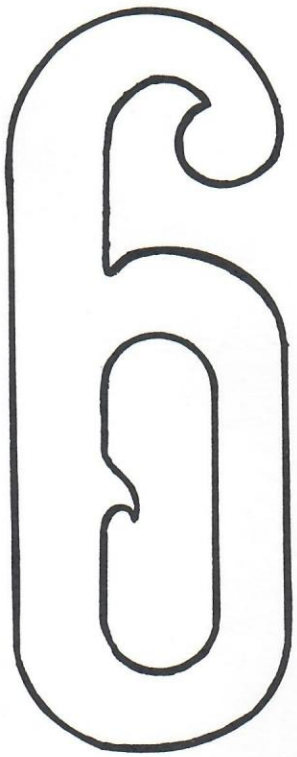


A.C. LOAD VOLTAGE

# Ed-Lab

## *EXPERIMENT MANUAL*

### TRANSISTORS



**UNIT NO.**

CES INDUSTRIES, INC.

Ed-Lab

UNIT NO. 6

# EXPERIMENT MANUAL

# TRANSISTORS

Copyright © 1974, 1976, 1980, 1981 CES INDUSTRIES, INC.

Name_____

Course_____

TABLE OF CONTENTS


1. NPN Transistor
2. Common Emitter D.C. Amplifier
3. Transistor and Beta Tester
4. Common Emitter Amplifier Biasing
5. A.C. Common Emitter Amplifier
6. C.E. Input and Output Impedance
7. Common Collector Amplifier or Emitter Follower
8. E.F. Input and Output Impedance
9. Emitter Bias and Degeneration
10. Emitter By-Pass Capacitor
11. Dynamic Amplifier and Phase Splitter
12. Transistor Characteristic Curve Tracer
13. Common Base Amplifier
14. Common Base A.C. Amplifier
15. Cascaded Amplifier
16. PNP Transistor
17. NPN - PNP Cascaded Amplifier
18. Constant Current Source
19. Difference Circuit
20. Difference Amplifier
21. Constant Current Difference Amplifier
22. Darlington Amplifier
23. Audio Driver
24. Feedback Amplifier
25. FET Operation
26. FET Common Source Amplifier
27. FET Common Drain Amplifier
28. FET Common Gate Amplifier
29. FET Characteristic
30. FET Constant Current Source

TABLE OF CONTENTS

1. NPN Translator
2. Common Emitter D.C. Amplifier
3. Translator and Data Tester
4. Common Emitter Amplifier Biasing
5. A.C. Common Emitter Amplifier
6. C.E. Input and Output Impedance
7. Common Collector Amplifier or Emitter Follower
8. E.F. Input and Output Impedance
9. Emitter Bias and Degeneration
10. Emitter By-Pass Capacitor
11. Dynamic Amplifier and Phase Splitter
12. Transistor Characteristic Curve Tracer
13. Common Base Amplifier
14. Common Base A.C. Amplifier
15. Cascaded Amplifier
16. PNP Translator
17. NPN - PNP Cascaded Amplifier
18. Complete Control System
19. Multivibrator Circuit
20. Differential Amplifier
21. Constant Current Difference Amplifier
22. Darlington Amplifier
23. Audio Driver
24. Feedback Amplifier
25. FET Operation
26. FET Common Source Amplifier
27. FET Common Drain Amplifier
28. FET Common Gate Amplifier
29. FET Characteristic
30. FET Constant Current Source

**Ed-Lab** | NPN TRANSISTOR | **EXP. NO. 1** **Page No. 1**

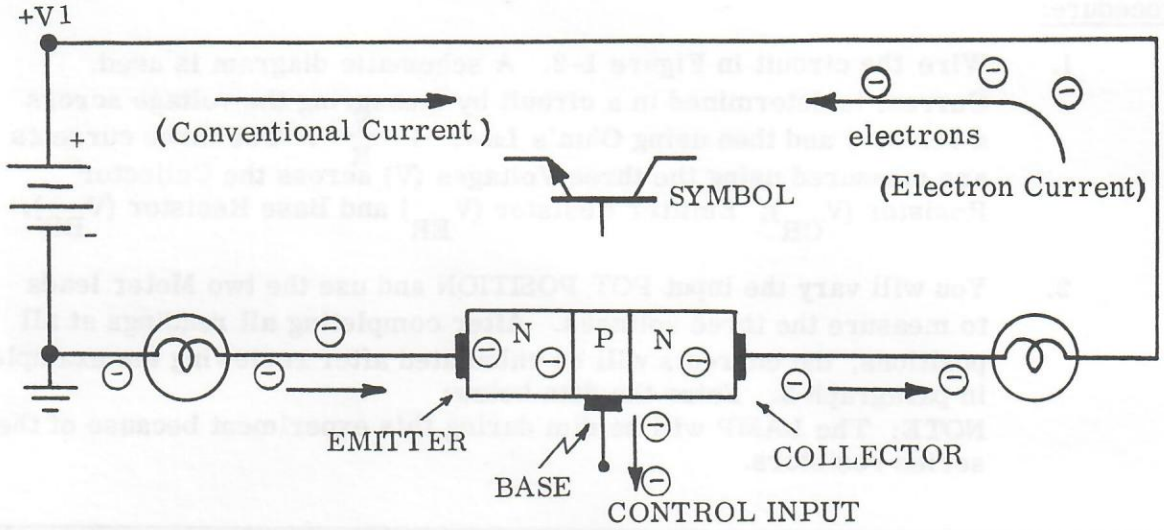



Figure 1 - 1 Transistor Electron Flow Control

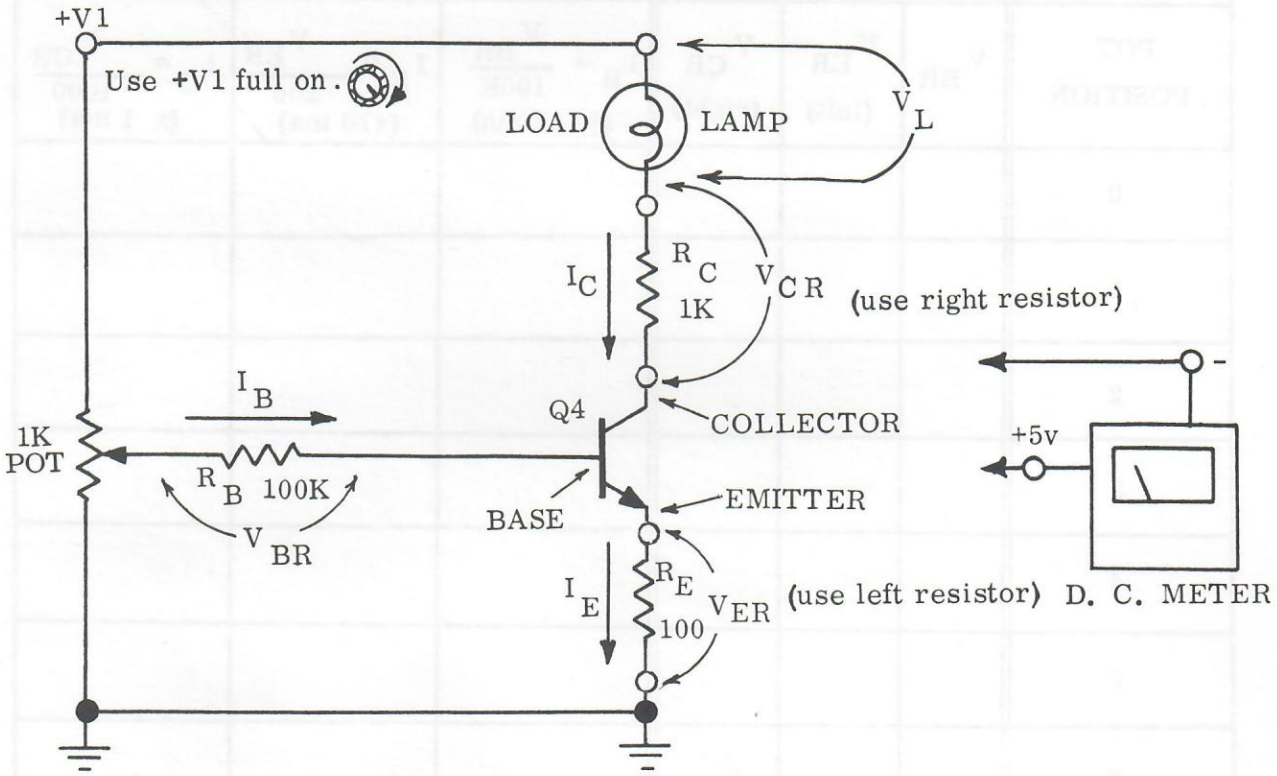



Figure 1 - 2 NPN Transistor Circuit

Objective:

You will use an NPN TRANSISTOR in a circuit where a control Input is applied to the BASE to control the flow of CURRENT or ELECTRONS through a LOAD or LAMP. The input BASE current and the controlled COLLECTOR current and EMITTER current are measured.

| Ed-Lab | NPN TRANSISTOR | EXP. NO. 1<br>Page No. 2 |
|---|---|---|

Procedure:

1. Wire the circuit in Figure 1-2. A schematic diagram is used. Current is determined in a circuit by measuring the voltage across a resistor and then using Ohm's Law; $I = \frac{V}{R}$. The three currents are measured using the three Voltages (V) across the Collector Resistor ($V_{CR}$); Emitter Resistor ($V_{ER}$) and Base Resistor ($V_{BR}$).

2. You will vary the input POT POSITION and use the two Meter leads to measure the three voltages. After completing all readings at all positions, the currents will be calculated after reviewing the examples in paragraph 3. Enter the data below:
NOTE: The LAMP will be dim during this experiment because of the series resistors.

| SET | READ | | | CALCULATE (see examples) | | |
|---|---|---|---|---|---|---|
| POT POSITION | $V_{BR}$ | $V_{ER}$ (left) | $V_{CR}$ (right) | $I_B = \frac{V_{BR}}{100K}$ (÷100 ma) | $I_E = \frac{V_{ER}}{100}$ (x10 ma) | $I_C = \frac{V_{CR}}{1000}$ (x 1 ma) |
| 0 | | | | | | |
| 1 | | | | | | |
| 2 | | | | | | |
| 3 | | | | | | |
| 4 | | | | | | |
| 6 | | | | | | |
| 8 | | | | | | |
| 10 | | | | | | |

| Ed-Lab | NPN TRANSISTOR | EXP. NO. 1<br>Page No. 3 |
|---|---|---|

3. Calculate the currents in the BASE, COLLECTOR, and EMITTER. Examples are:

(a) For $V_{BR} = 0.75$ volts

$$I_B = \frac{V_{BR}}{100K} = \frac{V_{BR}}{100} \text{ milliamps}$$

$$I_B = \frac{V_{BR}}{100} = \frac{0.75}{100} \text{ ma.}$$

$$I_B = \frac{0.75}{100} = 0.0075\text{ma}$$

(b) For $V_{ER} = 2.25$ volts

$$I_E = \frac{V_{ER}}{100} = 10\ V_{ER} \text{ milliamps}$$

$$I_E = 10\ V_{ER}$$

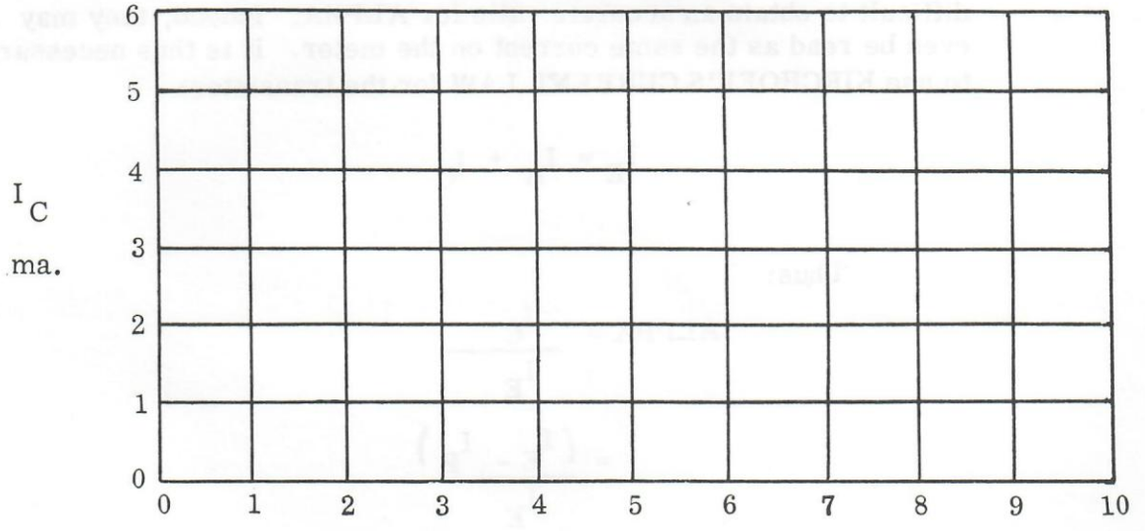
$$I_E = 10 \times 2.25 = 22.5 \text{ ma.}$$

Enter the calculated currents (just shift the voltage decimal points) in the table on the previous page.

4. How do the Emitter and Collector Currents compare?

5. How do the Base and Collector Currents compare?

6. Plot the Collector Current versus the Pot Position below:



7. Why does the collector current "level-off" or "limit"?

8. Current Gains

The OPERATING OR LINEAR RANGE of the circuit is in the sloped portion of the curve. The current gains can be calculated in this range.

The ratio of the Collector Current to the Base Current is called BETA ($\beta$), which is the Greek letter for (B).

Thus:

$$\text{BETA } (\beta) = \frac{I_C}{I_B}$$

The ratio of the Collector Current to the Emitter Current is called (ALPHA ($\alpha$), which is the Greek letter for (A).

Thus:

$$\text{ALPHA } (\alpha) = \frac{I_C}{I_E}$$

9. Calculate BETA from a point on the linear portion of the curve. Use POT position 3.

$$\text{TRANSISTOR } \beta = \frac{I_C}{I_B} =$$

$$\beta =$$

10. The Collector and Emitter Currents are almost equal so that it is difficult to obtain an accurate ratio for ALPHA. Indeed, they may even be read as the same current on the meter. It is thus necessary to use KIRCHOFF'S CURRENT LAW for the transistor:

$$I_E = I_B + I_C$$

Thus:

$$\text{ALPHA} = \frac{I_C}{I_E}$$

$$= \frac{(I_E - I_B)}{I_E}$$

$$\text{ALPHA } (\alpha) = 1 - \left( \frac{I_B}{I_E} \right)$$

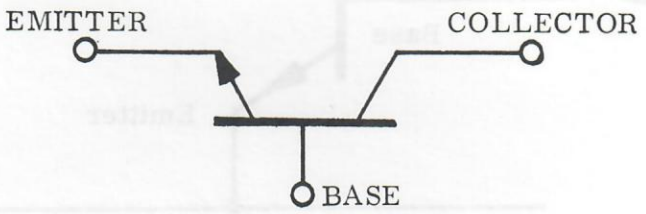
Calculate ALPHA

$$\alpha = 1 - \left( \frac{I_B}{I_E} \right) =$$

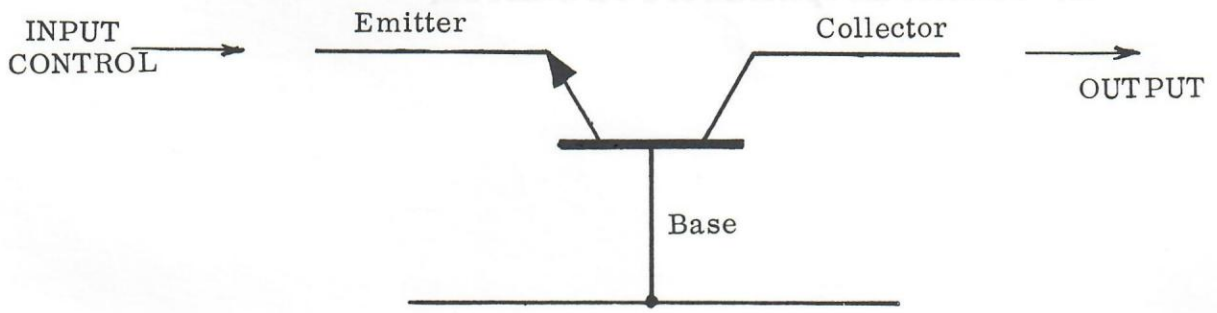
$\alpha =$

11. If a Transistor has a BETA of 50, and it is desired to operate a LOAD in the COLLECTOR or EMITTER of 0.25 amperes, how much current must be provided into the BASE?

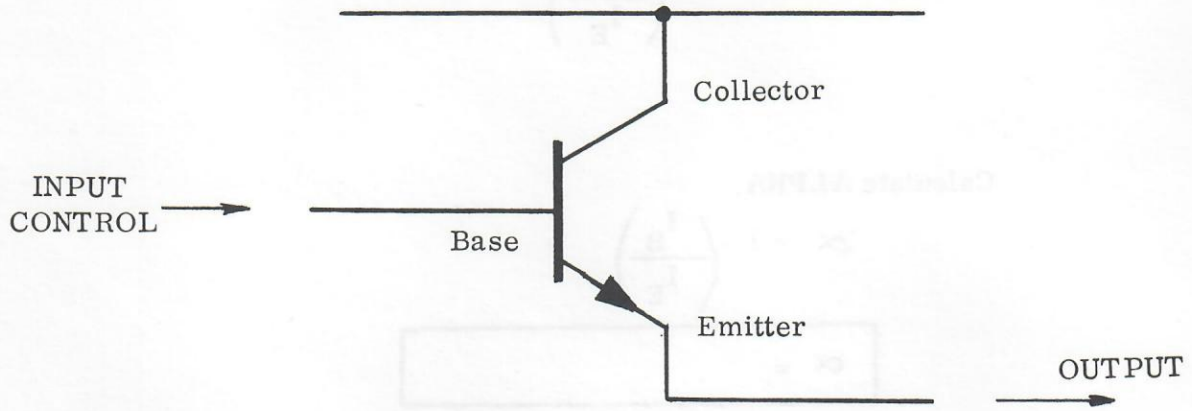
12. The transistor has 3 terminals:



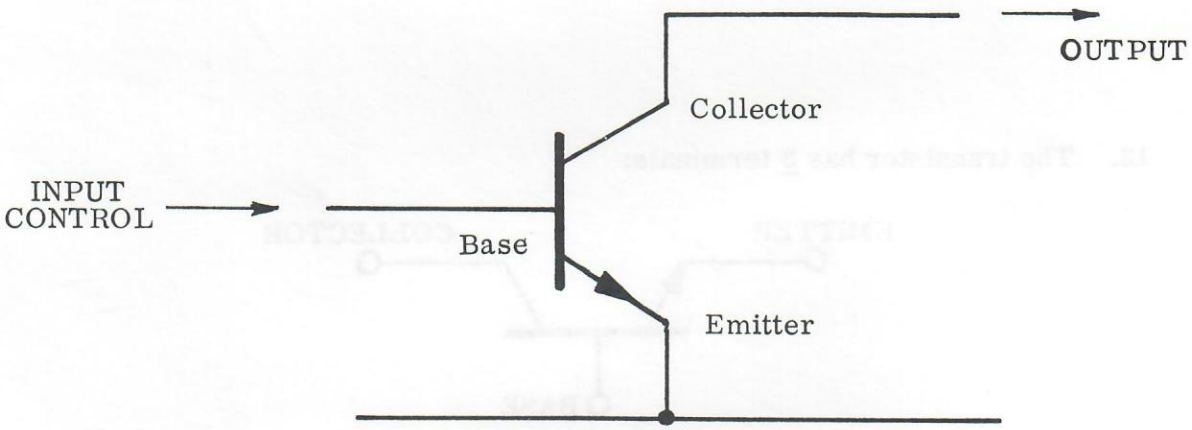

Three different combinations of input and output terminals can be used with a transistor as shown below and on the following page.




COMMON BASE




COMMON COLLECTOR




COMMON EMITTER

13. Describe the operation of a TRANSISTOR.

**Ed-Lab** | COMMON EMITTER D.C. AMPLIFIER | **EXP. NO.** 2 **Page No.** 1

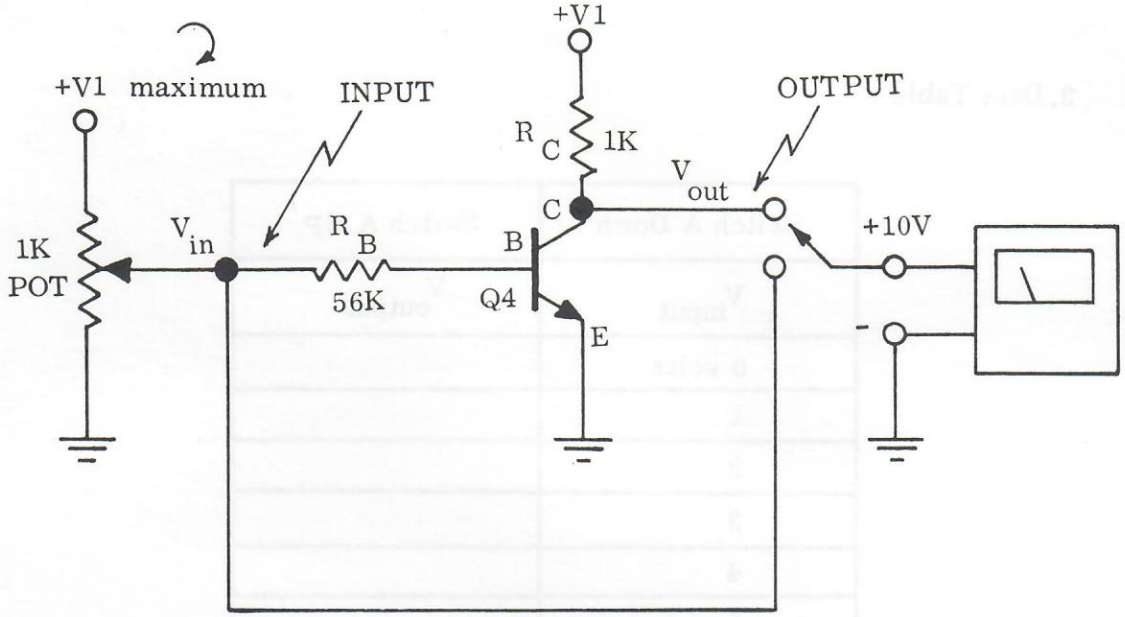


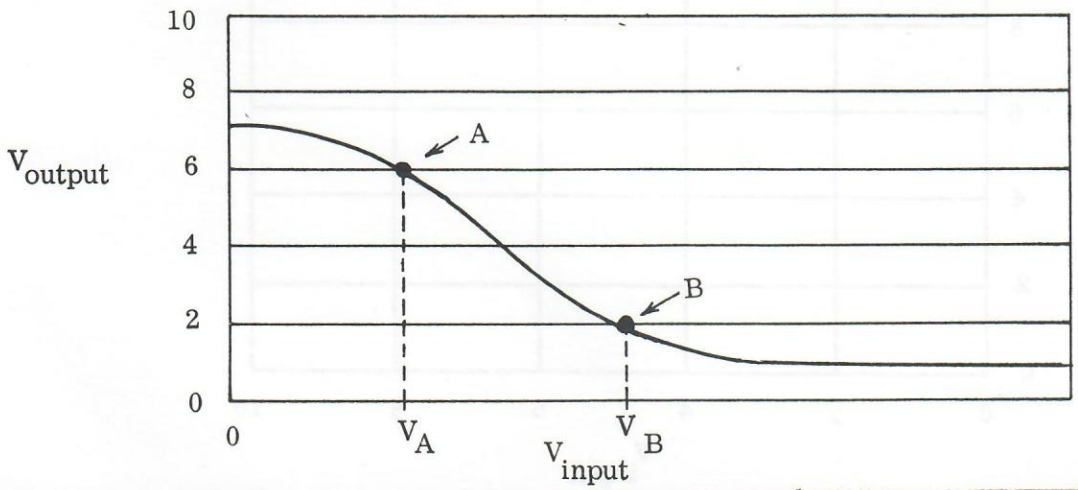

Figure 2 - 1 Common Emitter Amplifier

Objective:

You will apply an Input D.C. Voltage to the BASE of the COMMON EMITTER Amplifier Circuit with the Output Voltage at the COLLECTOR. The Input-Output TRANSFER CURVE is determined and you will calculate the AMPLIFIER GAIN, the OPERATING RANGE of the AMPLIFIER CIRCUIT, and the CURRENT GAIN (BETA) of the TRANSISTOR.

Procedure:

1. Wire the circuit. Set +V1 to maximum on all experiments.
2. Vary the INPUT (1K POT) and read the Input Voltages ( Switch A DOWN) and the Output Voltages (Switch A UP). Use the values shown in the table and enter the data in the table on the following page. Then plot the results on the graph. A typical graph is shown below:

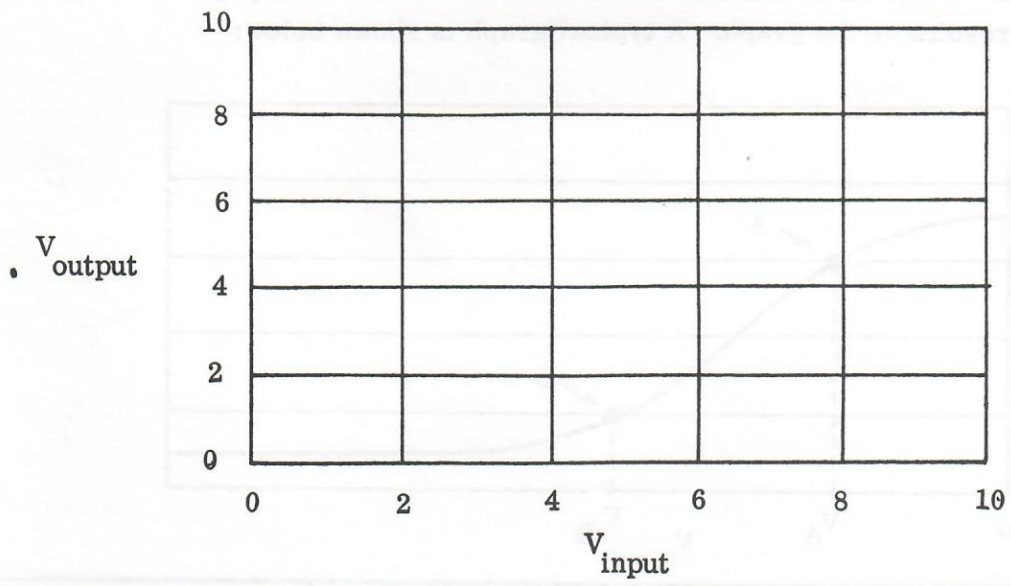


| Ed-Lab | COMMON EMITTER D.C. AMPLIFIER | EXP. NO. 2<br>Page No. 2 |
|---|---|---|

3. Data Table

| Switch A Down | Switch A UP |
|---|---|
| $V_{input}$ | $V_{output}$ |
| 0 volts | |
| 1 | |
| 2 | |
| 3 | |
| 4 | |
| 5 | |
| 6 | |
| 7 | |
| 8 | |
| | |
| | |

4. Input-Output graph



a. The "down-slope" shows that the output voltage goes DOWN as the input voltage goes UP. This means that the AMPLIFIER GAIN (G) is negative or has a Phase Inversion as shown later in A.C. Amplifiers.

b. The AMPLIFIER GAIN (G) is equal to the slope or the ratio of change in output voltage to change in input voltage which is calculated using any two points on the curve (A and B as shown on the sketch on page one). A calculation is given below for points corresponding to output voltages of 6v. and 2v.

$$\text{GAIN} = \frac{(V_{\text{out B}} - V_{\text{out A}})}{(V_{\text{in B}} - V_{\text{in A}})}$$

$$G = \frac{(2 - 6)}{(V_B - V_A)}$$

$$G = - \frac{4}{(V_B - V_A)}$$

Using your curve in Paragraph 4 above, calculate the amplifier gain, G.

$$\text{GAIN} = - \frac{4}{(V_B - V_A)} =$$

| GAIN (G) = — |
|---|

5. The sloping or linear portion of the curve only covers a small portion of the Input Voltage range. What is this LINEAR OPERATING RANGE for the Input Voltage variations on the curve in Paragraph 4.

| INPUT VOLTAGE OPERATING RANGE |
|---|
| $V_{input}$ MINIMUM = |
| $V_{input}$ MAXIMUM = |
| Range (MAX-MIN) = |

6. TRANSISTOR CURRENT GAIN CALCULATIONS:

The input BASE current, $I_B$, is:

$$I_B = \frac{V_{in}}{R_B}$$

The COLLECTOR current, $I_C$, is determined by the transistor gain BETA or $\beta$ (Greek B).

$$I_C = \beta \ I_B$$

The Output Voltage (change) $V_{out}$* is given by the Collector Current times the collector Resistor and is:

$$V_{out}^* = I_C R_C$$

Substituting $I_C = \beta \cdot I_B$, and then $I_B = \frac{V_{in}}{R_B}$, we obtain:

$$V_{out}^* = (\beta \cdot I_B) \cdot R_C$$

and $$V_{out}^* = \beta \cdot \left(\frac{V_{in}}{R_B}\right) \cdot R_C$$

Solving for the Amplifier Gain (G) or $\left(\frac{V_{out}^*}{V_{in}}\right)$ we obtain:

$$G = \frac{V_{out}^*}{V_{in}} = \beta \frac{R_C}{R_B}$$

We can also solve for the transistor gain, $\beta$, from the above equation.

$$\beta = (G) \frac{R_B}{R_C}$$

Using the Gain (G) calculated in paragraph 4 above, calculate the Transistor $\beta$

$$\beta = (G) \frac{R_B}{R_C}$$

$$\beta = (G) \left(\frac{56}{1}\right) =$$

| $\beta$ = |
|---|

7. Amplifier GAIN can now be calculated for any size resistors. Consider:

$$R_C = 2.2K$$

$$R_B = 100K$$

The Amplifier Gain ($G_2$) is:

$$G_2 = -\beta \frac{R_C}{R_B}$$

$$G_2 = -\beta \frac{2.2}{100} = \frac{-\beta}{45.45}$$

Substitute $\beta$ calculated in Paragraph 6 above

$$G_2 = -\frac{\beta}{45.45}$$

$$G_2 = -$$

8. Use the same circuit as in Figure 2-1. Substitute these resistors:

$R_C$ = 2.2K for 1K

$R_B$ = 100K for 56K

The Amplifier Gain ($G_2$) was calculated above. You will measure the actual Gain (K) using only 2 points or measurements. Adjust the Output Voltage to 2 and 6 volts and measure the corresponding Input Voltages. Enter the data below:

| | set | read |
|---|---|---|
| POINT | Switch A UP | Switch A DOWN |
| | $V_{output}$ | $V_{input}$ |
| A | 2 | $V_A$ = |
| B | 6 | $V_B$ = |

Calculate this measured gain, K.

$$K = \frac{(V_{out\ B} - V_{out\ A})}{(V_{in\ B} - V_{in\ A})}$$

$$K = \frac{4}{(V_B - V_A)}$$

| K = — |
|---|

9. How does your CALCULATED GAIN, K, compare with your measured gain, $G_2$, above?

10. What is the amplifier gain for a transistor with a $\beta$ of 80 and $R_B$ = 100K and $R_C$ = 3.3K?

11. How does the $\beta$ measured in this experiment compare with $\beta$ calculated in Experiment No. 1?

**Ed-Lab** | TRANSISTOR & BETA TESTER | **EXP. NO.** 3 **Page No.** 1

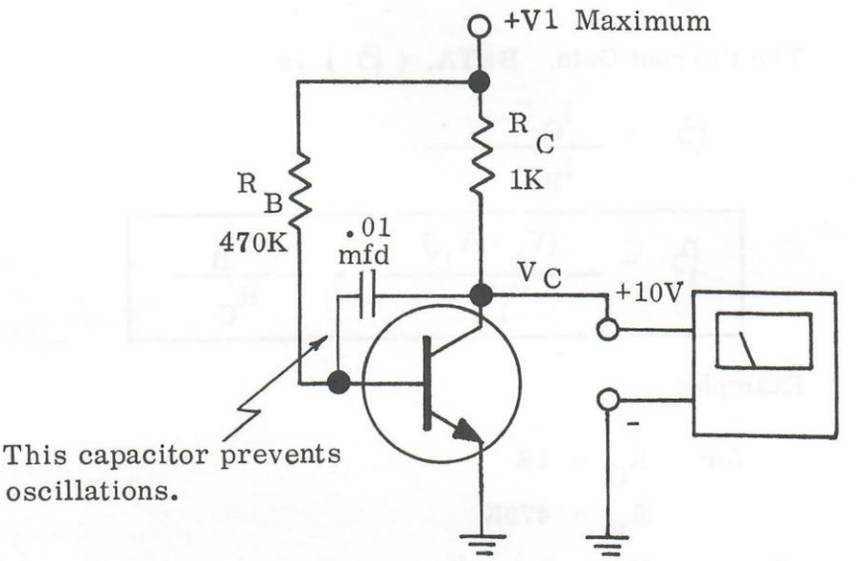



Figure 3 - 1 Transistor Beta Tester

Objective:

You will connect Base and Collector Resistors to provide a Collector Voltage in the Linear Region of Operation. The current Gain or BETA is then calculated from measurement.

Description:

A base resistor, $R_B$, is inserted until the collector voltage reads some mid value of $V_C$. The Base Current is:

$$I_B = \frac{V_1}{R_B}$$

The Collector Current is:

$$I_C = \frac{(V_1 - V_C)}{R_C}$$

The Current Gain, BETA, ( $\beta$ ) is:

$$\beta = \frac{I_C}{I_B}$$

$$\boxed{\beta = \frac{(V_1 - V_C)}{V_1} \cdot \frac{R_B}{R_C}}$$

Example:

for $R_C = 1K$

$R_B = 470K$

$V_1 = 9.1$ volts

$V_C = 5.5$ volts

$$\beta = \frac{(9.1 - 5.5)}{9.1} \cdot \frac{470}{1}$$

$$= \frac{3.6}{9.1} \cdot \frac{470}{1}$$

$$\beta = 186$$

Procedure:

1. Wire the circuit shown using transistor Q4. Insert various values of $R_B$ until any mid-value of $V_C$ is achieved, say between 4 and 6 volts. Record the value of $V_1$ $V_C$, and $R_B$ for each different NPN transistor, Q4, Q5, Q6 and Q7. Transistor Q7 is a SILICON transistor and the others are GERMANIUM.

| Ed-Lab | TRANSISTOR & BETA TESTER | EXP. NO. 3<br>Page No. 3 |
|---|---|---|

2. Enter the readings below, and then calculate $\beta$.

| | data | | | |
|---|---|---|---|---|
| TRANSISTOR | $V_C$ | $R_B$ | +V1 | CALCULATE BETA |
| Q4 | | | | |
| Q5 | | | | |
| Q6 | | | | |
| Q7 | | | | |

**Ed-Lab** | COMMON EMITTER AMPLIFIER BIASING | **EXP. NO.** 4 **Page No.** 1

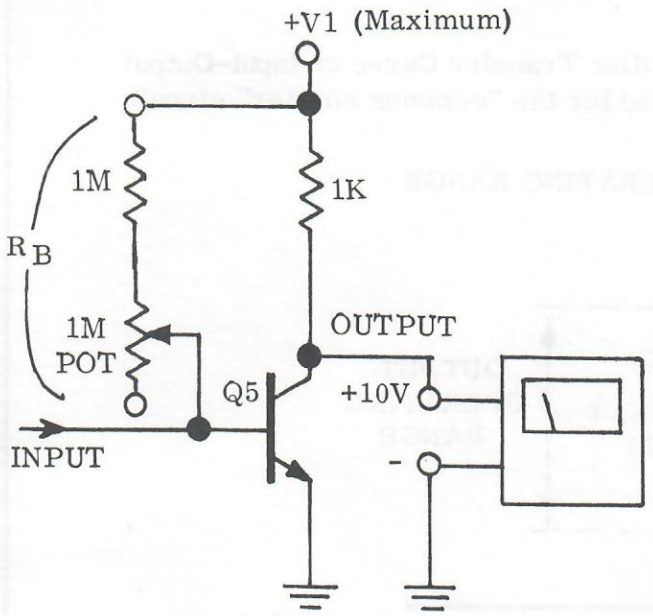



Figure 4 - 1 Small Signal Bias

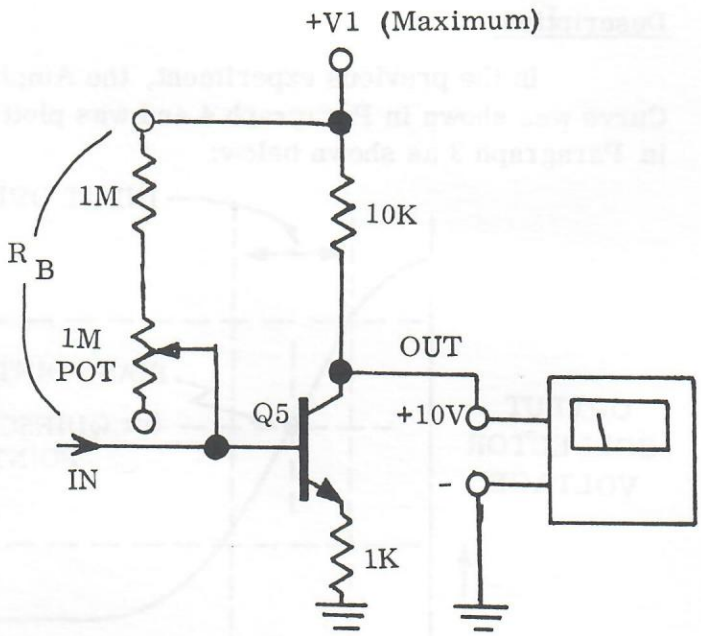



Figure 4 - 2 Emitter Bias

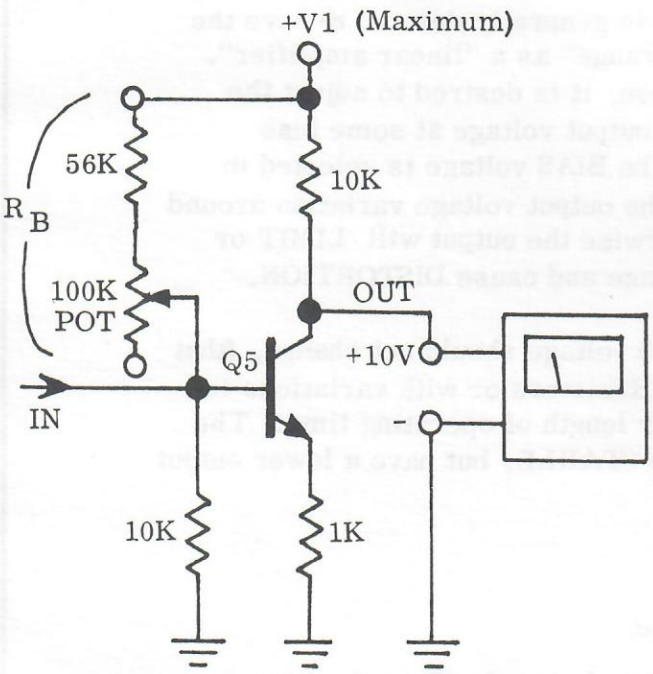



Figure 4 - 3 Voltage Divider Bias

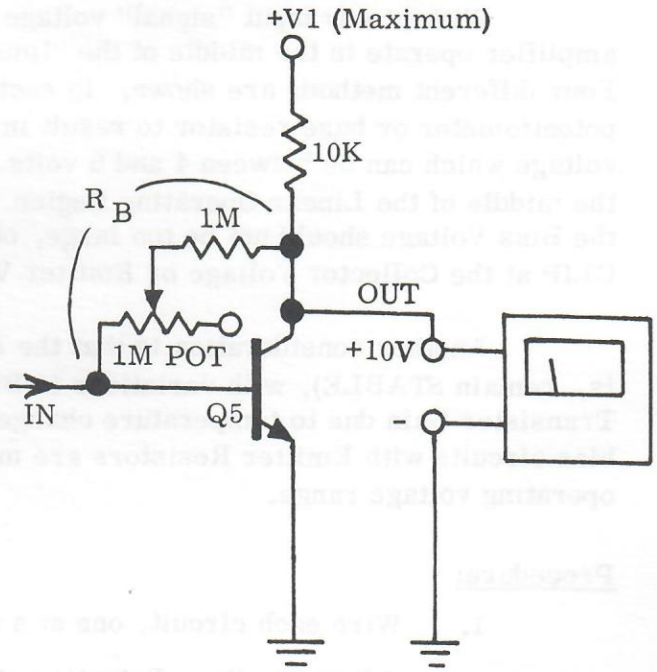



Figure 4 - 4 Collector Feedback Bias

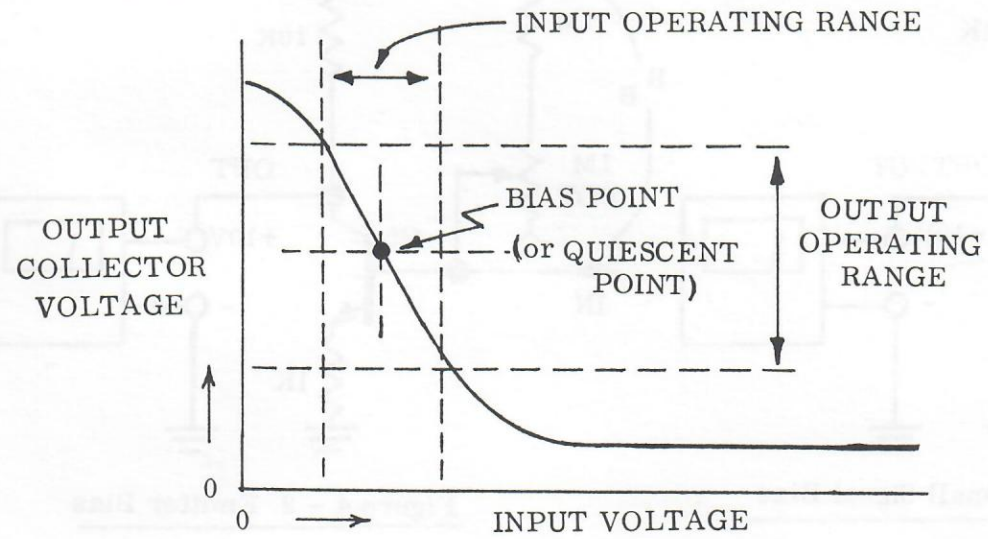
Objective:

You will use resistors in the BASE of a common emitter amplifier to provide an OPERATING POINT BIAS in the LINEAR operating region of the AMPLIFIER.

| Ed-Lab | COMMON EMITTER AMPLIFIER BIASING | EXP. NO. 4<br>Page No. 2 |
|---|---|---|

Description:

In the previous experiment, the Amplifier Transfer Curve or Input-Output Curve was shown in Paragraph 4 and was plotted for the "common emitter" circuit in Paragraph 3 as shown below:




Without any input "signal" voltage, it is generally desired to have the amplifier operate in the middle of the "linear range" as a "linear amplifier". Four different methods are shown. In each case, it is desired to adjust the potentiometer or base resistor to result in an output voltage at some bias voltage which can be between 4 and 6 volts. The BIAS voltage is selected in the middle of the Linear Operating Region. The output voltage variation around the Bias Voltage should not be too large, otherwise the output will LIMIT or CLIP at the Collector Voltage or Emitter Voltage and cause DISTORTION.

Another consideration is that the BIAS voltage should not change, (that is, remain STABLE), with variations in Bias Resistors or with variations in Transistor Gain due to temperature changes or length of operating time. The bias circuits with Emitter Resistors are more STABLE, but have a lower output operating voltage range.

Procedure:

1. Wire each circuit, one at a time.

   Adjust the Base Potentiometer or change the Base Resistor in each BIAS circuit to result in an output bias of between 3 and 7 volts and in the linear range. Estimate the Base Resistor.

   Substitute Q4, Q6 and Q7 and comment on the STABILITY of the output voltage level.

Enter your comments in the table below:

| Figure | Circuit | TRANSISTOR Q5 | | $V_C$ (STABILITY) | | |
|---|---|---|---|---|---|---|
| | | $V_C$ (output) | $R_B$ (estimated) | Q4 | Q6 | Q7 |
| 4 - 1 | Small Signal Bias | | | | | |
| 4 - 2 | Emitter Bias | | | | | |
| 4 - 3 | Voltage Divider Bias | | | | | |
| 4 - 4 | Collector Feedback Bias | | | | | |

2. Other BIAS Circuits are shown below:

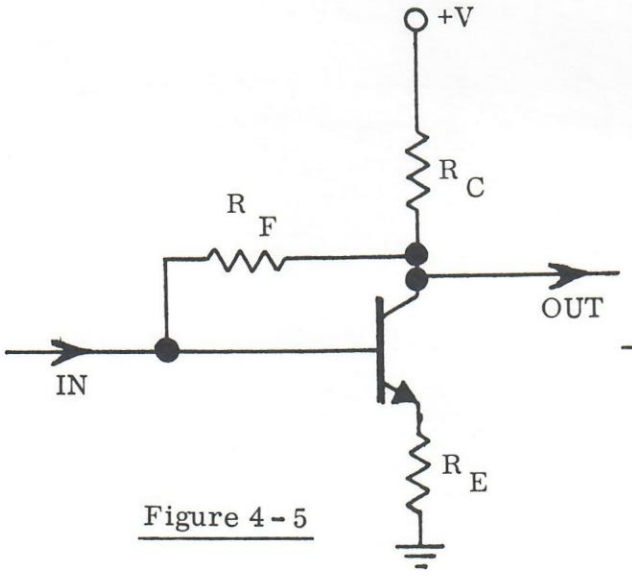



Figure 4 - 5

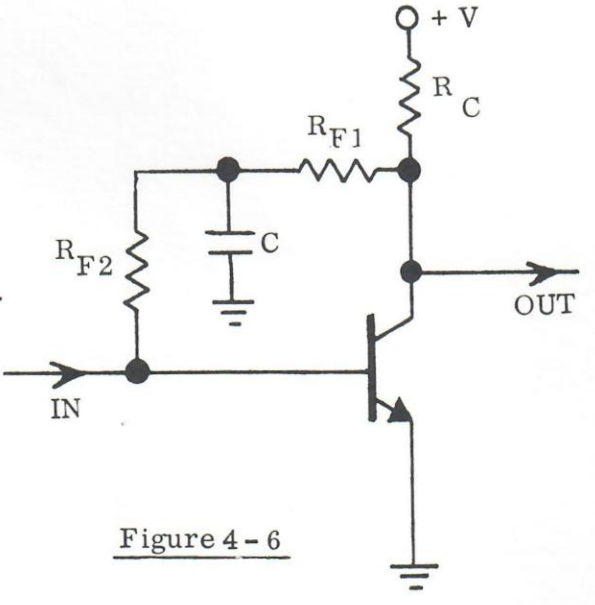



Figure 4 - 6

Fig. 4 - 5. The Collector Feedback can also be combined with an Emitter Resistor for added stability.

Fig. 4 - 6. The Feedback Resistor can be separated with a Capacitor connected to ground. This permits D. C. Feedback for Bias Stability but eliminating any A. C. feedback which would otherwise reduce gain.

| Ed-Lab | A.C. COMMON EMITTER AMPLIFIER | EXP. NO. 5<br>Page No. 1 |
|---|---|---|

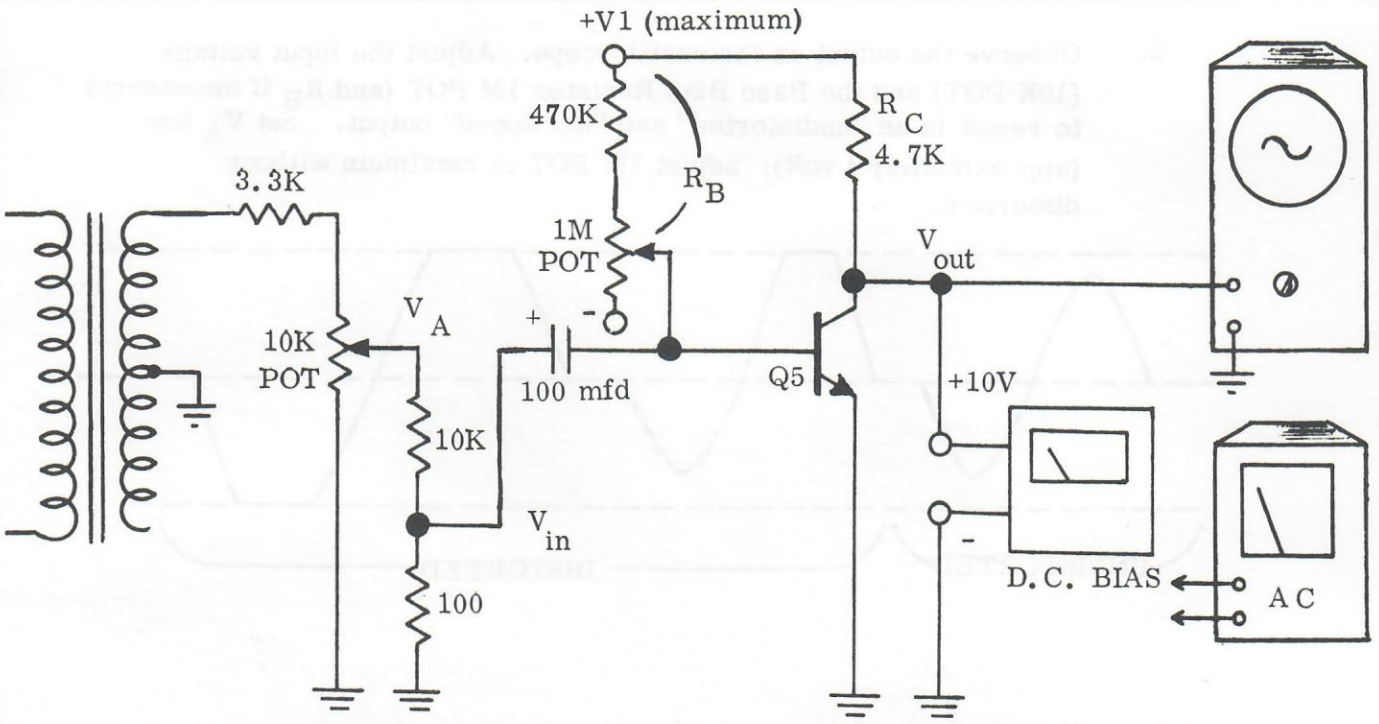



Figure 5 - 1 A.C. Amplifier

Objective:

You will use a common emitter amplifier with an A.C. Capacitor Coupling in an A.C. Amplifier Circuit and measure the Gain.

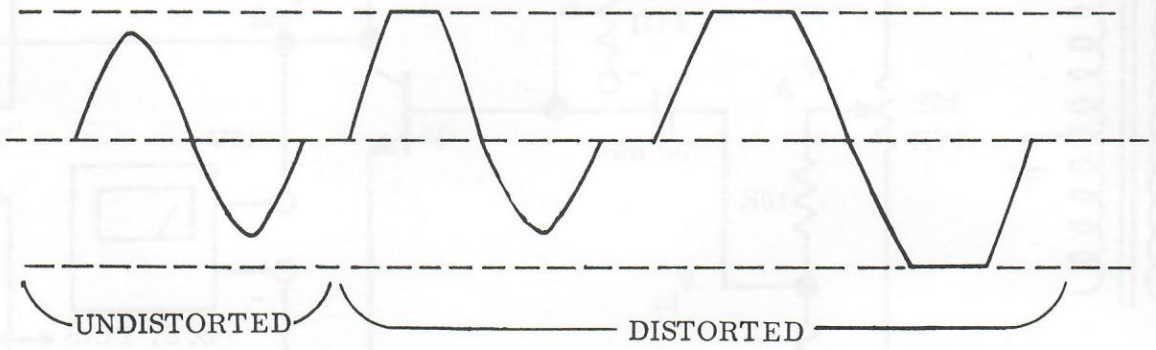
Procedure:

1. Wire the circuit. The transformer is used to provide an A.C. signal and the 10K Potentiometer is used to vary the signal amplitude. The 10K and 100Ω voltage divider is used to provide a small input signal. The larger signal, $V_A$ can be easily measured on the A.C. Multimeter. The input voltage, $V_{in}$, is then:

$$V_{in} = V_A \frac{100}{10,100}$$

$$V_{in} = \frac{V_A}{100}$$ Approximation

2. Observe the output on the oscilloscope. Adjust the input voltage (10K POT) and the Base Bias Resistor 1M POT (and $R_B$ if necessary) to result in an "undistorted" and "unclipped" output. Set $V_A$ low (approximately 1 volt); adjust 1M POT to maximum without distortion.




3. Use the A.C. Meter or the Oscilloscope to read the input A.C. Voltage and output A.C. Voltages (RMS on the meter, or peak to peak on the 'scope).

$V_A$ = _____ volts

$V_{in} = \frac{V_A}{100}$ = _____ volts

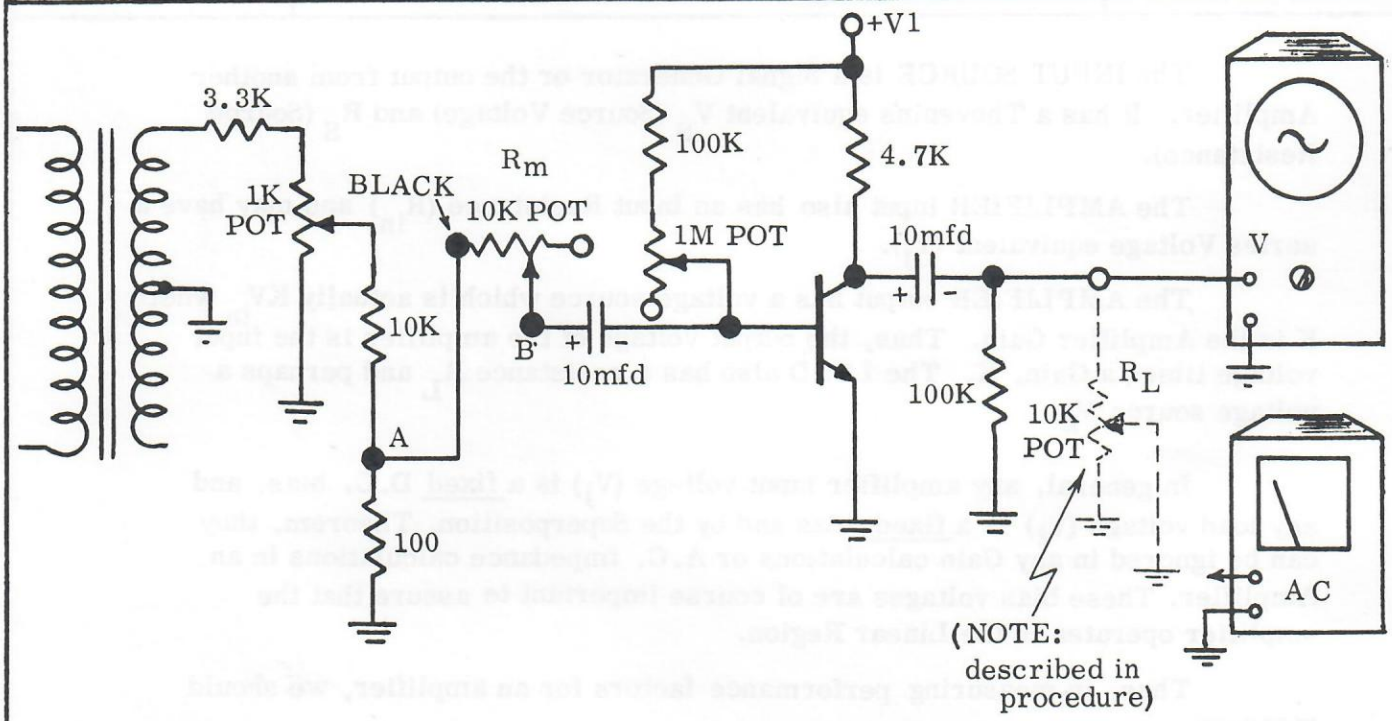
$V_{out}$ = _____ volts

4. Calculate the amplifier gain, G,

$$G = \frac{V_{out}}{V_{in}} =$$

| G = |
|---|

**Ed-Lab** | COMMON EMITTER INPUT and OUTPUT IMPEDANCE | EXP. NO. 6 Page No. 1

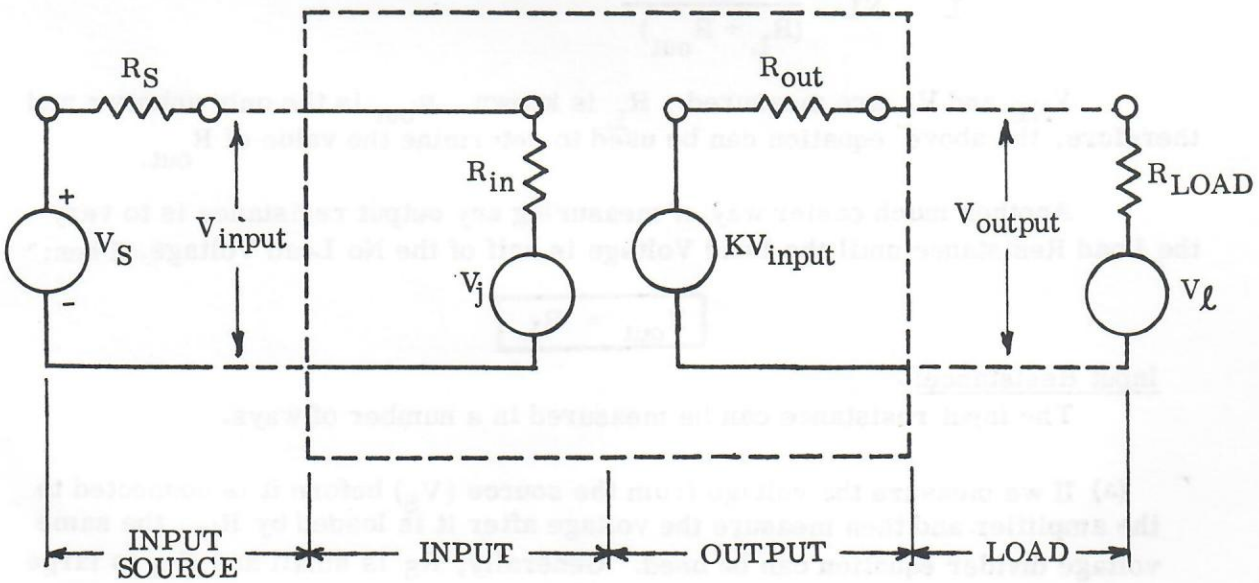



Objective:

You will measure the internal Input and Output Resistances or Impedances of the Common Emitter Amplifier.

Description:

Thevenin's Theorem in D.C. Circuit Theory states that the "equivalent" circuit between any two points in a circuit can be described as a simple resistor in series with a voltage source. A general simplified circuit for any component or any transistor or any amplifier is shown below:



The INPUT SOURCE is a Signal Generator or the output from another Amplifier. It has a Thevenin's equivalent $V_S$ (Source Voltage) and $R_S$ (Source Resistance).

The AMPLIFIER input also has an Input Resistance ($R_{in}$) and may have a series Voltage equivalent ($V_j$).

The AMPLIFIER output has a voltage source which is actually $KV_{in}$ where K is the Amplifier Gain. Thus, the output voltage of the amplifier is the input voltage times a Gain, K. The LOAD also has a resistance $R_L$ and perhaps a voltage source $V_\ell$.

In general, any amplifier input voltage ($V_j$) is a fixed D.C. bias, and any load voltage ($V_\ell$) is a fixed bias and by the Superposition, Theorem, they can be ignored in any Gain calculations or A.C. impedance calculations in an Amplifier. These bias voltages are of course important to assure that the amplifier operates in the Linear Region.

Thus, in measuring performance factors for an amplifier, we should measure:

$K$ , Gain
$R_{in}$ , Input Resistance
$R_{out}$, Output Resistance

It is generally desired to have a large $R_{in}$ and a small $R_{out}$.

Output Resistance:

Output Resistance is measured by reading the output voltage without a load ($V_{NL}$) and with a load ($V_L$). Then, with a Load of $R_L$;

$$V_L = V_{NL} \frac{R_L}{(R_L + R_{out})}$$

$V_{NL}$ and $V_L$ are measured. $R_L$ is known. $R_{out}$ is the only unknown and therefore, the above equation can be used to determine the value of $R_{out}$.

Another much easier way of measuring any output resistance is to vary the Load Resistance until the Load Voltage is half of the No Load Voltage. Then:

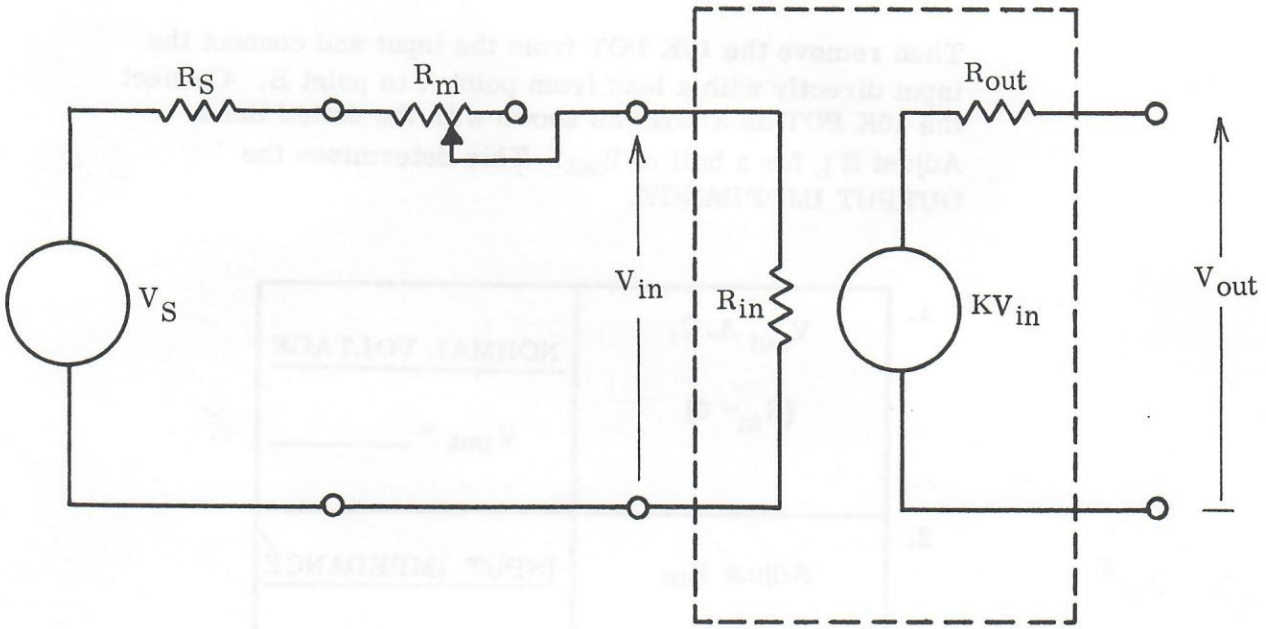
$$\boxed{R_{out} = R_L}$$

Input Resistance:

The input resistance can be measured in a number of ways.

(a) If we measure the voltage from the source ($V_S$) before it is connected to the amplifier and then measure the voltage after it is loaded by $R_{in}$, the same voltage divider equation can be used. Generally, $R_S$ is small and $R_{in}$ is large and the input voltage change is so small that $R_{in}$ cannot be measured accurately.

(b) The measuring method consists of inserting a special measuring resistor (large), $R_m$, in series with the input as shown below:




The voltage divider equation can then be used and $V_{in}$ can be measured using a high-impedance meter. When $V_{in}$ is reduced to half, and for small $R_S$, then

$$R_m = R_{in}$$

(c) Measuring $V_{in}$ can be a problem because the measuring meter must have a very high impedance not to load the circuit and $V_{in}$ can be very small and difficult to measure. We could measure $V_{out}$ using a scope or meter. Since $V_{out} = KV_{in}$, when $V_{out}$ is reduced to half, then $V_{in}$ is reduced to half and the same equation ($R_{in}=R_m$) applies. This is easier because the output voltage is higher and output resistance or impedance levels are smaller and therefore the measurements are easier.

Procedure:

1. Wire the circuit in Figure 6-1. Adjust the 10K POT to zero ohms.

2. Adjust the BIAS (1 MEG OHM POT) and the INPUT ( 1K OHM POT) for an output signal, viewed on the oscilloscope, without any distortion and in the linear region.

3. Adjust the input for a specific output as viewed on the oscilloscope and as read on the meter. Use the A.C. meter to determine "half"

output when determining input and output impedances.

4. Read $V_{out}$ and then vary the 10K POT ($R_m$) or until the output reads half of its initial value. This determines the INPUT IMPEDANCE.

   Then remove the 10K POT from the input and connect the input directly with a lead from point A to point B. Connect the 10K POT as a Load as shown with the dotted lines. Adjust $R_L$ for a half of $V_{out}$. This determines the OUTPUT IMPEDANCE.

| | | |
|---|---|---|
| 1. | $V_{out}$ A.C.<br>($R_m$= 0) | NORMAL VOLTAGE<br>$V_{out}$ = _____ |
| 2. | Adjust $R_m$<br>for 1/2 $V_{out}$<br>(1/2 $V_{out}$ = _____) | INPUT IMPEDANCE<br>$R_m = R_{in}$ = _____ |
| 3. | Remove $R_m$.<br>Add $R_L$.<br>Adjust $R_L$<br>for 1/2 $V_{out}$ | OUTPUT IMPEDANCE<br>$R_L = R_{out}$ = _____ |

**Ed-Lab** | COMMON COLLECTOR AMPLIFIER or EMITTER FOLLOWER | **EXP. NO. 7** **Page No. 1**

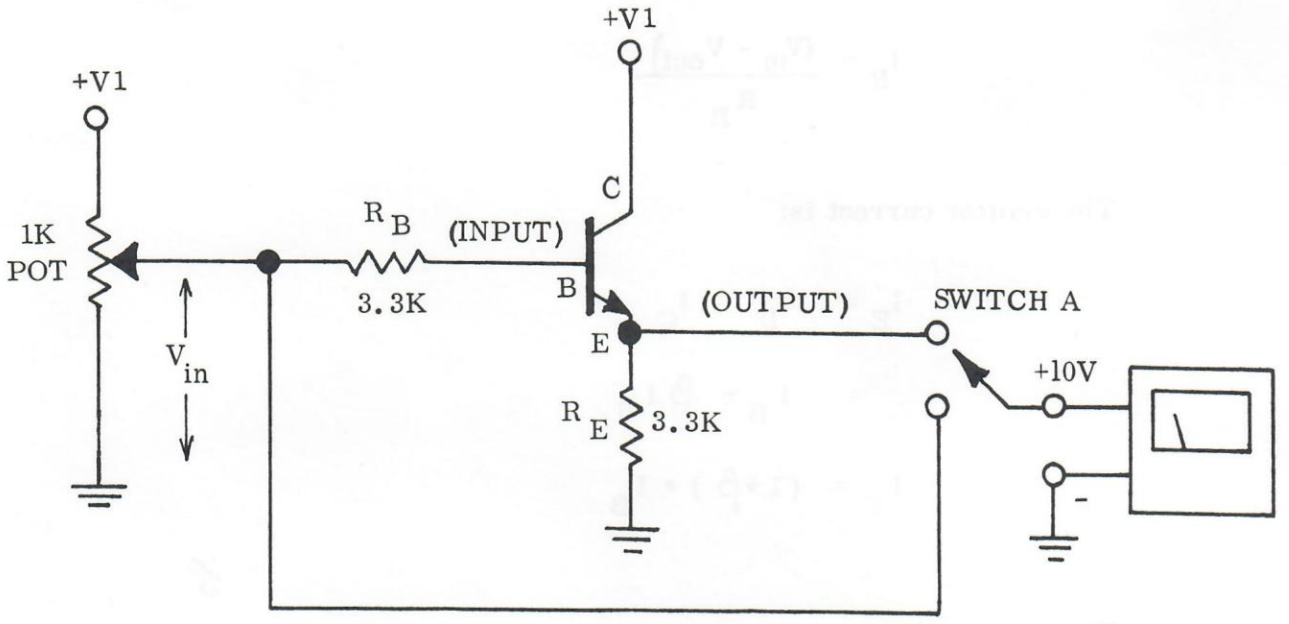



Figure 7 - 1 Emitter Follower

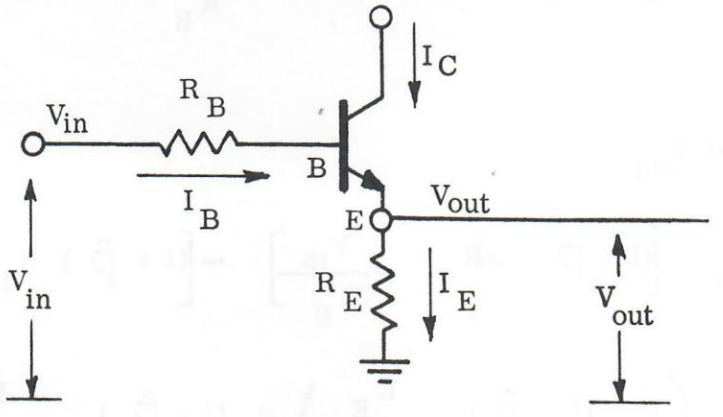
Objective:

You will apply an input to the transistor BASE and observe the output at the EMITTER in a Common Collector Amplifier Circuit and measure the Gain.

Description:

The Common Collector circuit is shown with the load current or voltage in the Emitter "leg" of the circuit.

The same transistor equations used in the other amplifiers apply.




$$I_C = \beta \; I_B$$

The base current is now:

$$I_B = \frac{(V_{in} - V_{out})}{R_B}$$

The emitter current is:

$$I_E = I_B + I_C$$

$$= I_B + \beta I_B$$

$$I_E = (1+\beta) \cdot I_B$$

The output voltage is:

$$V_{out} = R_E \cdot I_E$$

$$V_{out} = (1+\beta) \cdot R_E \cdot I_B$$

Substituting the value for Base Current:

$$V_{out} = (1+\beta)\ R_E\ \frac{(V_{in} - V_{out})}{R_B}$$

Solving for $V_{out}$:

$$V_{out} = \left[(1+\beta)\ R_E\ \frac{V_{in}}{R_B}\right] - \left[(1+\beta)\ R_E\ \frac{V_{out}}{R_B}\right]$$

$$V_{out}\left(1 + (1+\beta)\ \frac{R_E}{R_B}\right) = (1+\beta)\ \frac{R_E}{R_B}\ V_{in}$$

The gain is:

$$\text{GAIN} = \frac{V_{out}}{V_{in}} = \frac{(1+\beta)\,\dfrac{R_E}{R_B}}{1 + \dfrac{(1+\beta)\,R_E}{R_B}}$$

Consider the situation where $R_E = R_B$

then:

$$\text{Gain} = \frac{(1+\beta)}{1+(1+\beta)} = \frac{1+\beta}{2+\beta}$$

If $\beta = 100$, then

$$\text{Gain} = \frac{101}{102} \text{ or approximately 1.}$$

Observe that the Gain is always a little less than 1. Even if the $\beta = 25$, the gain is

$$\text{Gain} = \frac{26}{27}$$ or approximately 1. Indeed, the Gain will be approximately 1 with any $\beta$, and any $R_E$ and any $R_B$.

Procedure:

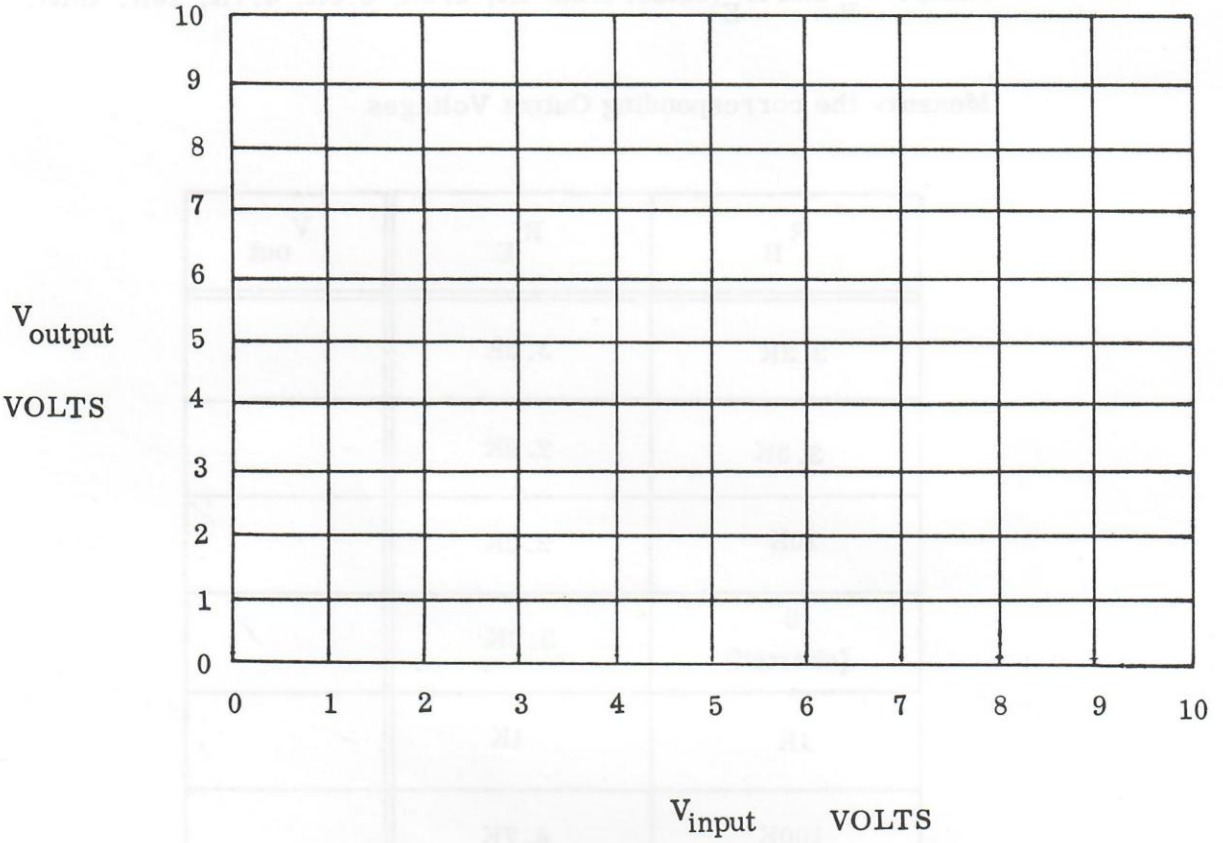
1. Wire the circuit in Figure 7 - 1.

2. Vary the input voltage and observe the output voltage and enter the data below:

| SWITCH A DOWN | SWITCH A UP |
|---|---|
| $V_{input}$ VOLTS | $V_{output}$ VOLTS |
| 0 | |
| 1 | |
| 2 | |
| 3 | |
| 4 | |
| 5 | |
| 6 | |
| 7 | |
| 8 | |
| 9 | |

| Ed-Lab | COMMON COLLECTOR AMPLIFIER or EMITTER FOLLOWER | EXP. NO. 7<br>Page No. 5 |
|---|---|---|

3. Plot the results on the graph below:




4. The Gain is the ratio of change in $V_{output}$ to change in $V_{input}$. Select any two points (from the table) and determine the gain of the common collector amplifier.

$$\text{GAIN} = \frac{(V_{outA} - V_{out\ B})}{(V_{in\ A} - V_{in\ B})} =$$

| GAIN = |
|---|

5. What is the GAIN of any Emitter Follower?

6. Set an input voltage corresponding to an output voltage of 5 volts.

Change $R_B$ and $R_E$ (select from 1K, 2.2K, 3.3K, 4.7K, 10K, 15K).

Measure the corresponding Output Voltages

| $R_B$ | $R_E$ | $V_{out}$ |
|---|---|---|
| 3.3K | 3.3K | |
| 3.3K | 2.2K | |
| 10K | 2.2K | |
| 0 (shorted) | 3.3K | |
| 1K | 1K | |
| 100K | 4.7K | |
| | | |

7. Comment on the variation in $V_{out}$ for different $R_B$ and $R_E$ resistors.

8. Why is this circuit called an EMITTER FOLLOWER?

| Ed-Lab | E. F. INPUT AND OUTPUT IMPEDANCE | EXP. NO. 8<br>Page No. 1 |
|---|---|---|

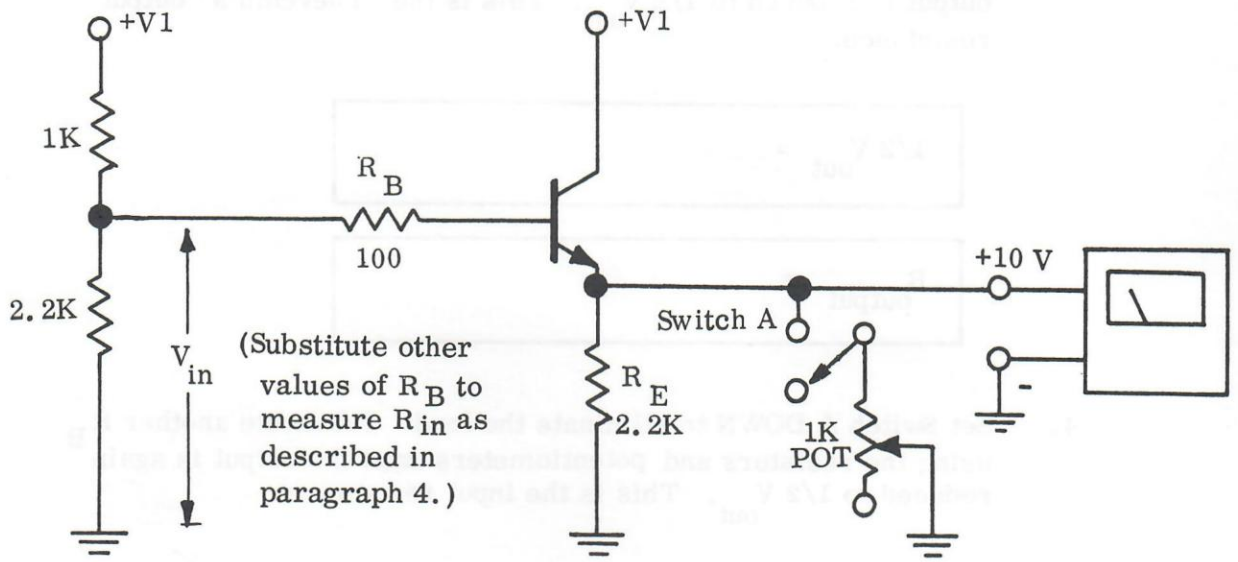



Figure 8-1 Impedance Measurement

Objective:

You will apply a series input resistor and an output load resistor to the Emitter Follower and determine the Input and Output Resistances of this circuit.

Description:

The previous experiment demonstrated that the output voltage was not changed with different input or output resistors. Therefore, this would indicate a large input resistance and a small output resistance which are desirable, so as not to load the input source, and not to be affected or loaded by an amplifier load. Recall that in the common emitter amplifier, the input and output impedances were small and there was a high gain.

Procedure:

1. Wire the circuit. Keep Switch A down to remove the load.
2. Read the output voltage

| $V_{out}$ = |
|---|

3. Set Switch A UP, to insert the LOAD POT and vary the load until the output is reduced to 1/2 $V_{out}$. This is the "Thevenin's" output resistance.

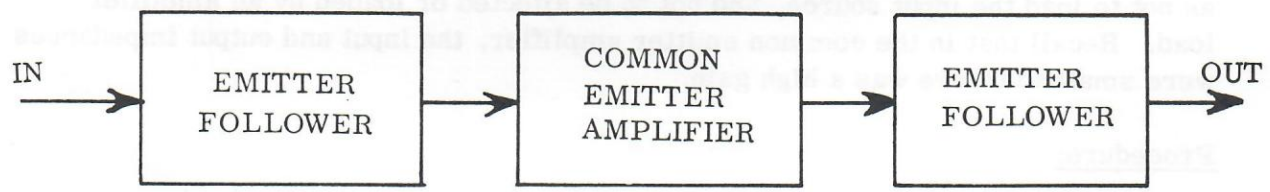
| $1/2\ V_{out}$ = |
|---|

| $R_{output}$ = |
|---|

4. Set Switch A DOWN to eliminate the load. Substitute another $R_B$ using the resistors and potentiometers until the output is again reduced to 1/2 $V_{out}$. This is the input resistance.

| $R_{input}$ = |
|---|

5. If we desired to have an ideal amplifier, we might use an emitter follower input for high input impedance, followed by a common emitter amplifier for high gain, and followed by another common collector, emitter follower which will not load the amplifier and provide a low output impedance.




High Input Resistance — High Gain — Low Output Resistance

6. It is of interest to note, without the math proof, that the Theoretical Input and Output Emitter Follower Resistances are:

$$R_{output} = \frac{R_E}{\beta}$$

$$R_{input} = \beta \cdot R_E$$

| Ed-Lab | EMITTER BIAS and DEGENERATION | EXP. NO. 9<br>Page No. 1 |
|---|---|---|

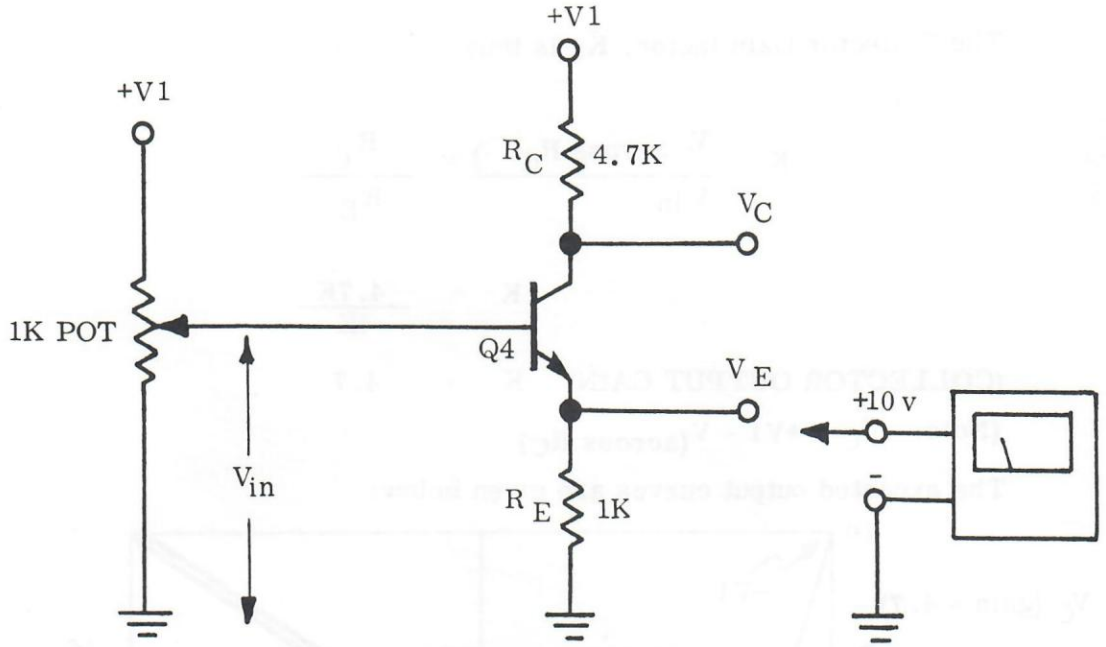



Figure 9-1 Emitter Degeneration

Objective:

You will vary the Input Voltage to a Transistor Amplifier with both a collector and emitter resistor and observe the operation and measure the gain.

Description:

The Emitter Resistor is in an Emitter Follower circuit. As in a previous experiment, the Emitter Voltage, $V_E$, is equal to the input voltage, $V_{in}$. The Emitter Current $I_E$ is then:

$$I_E = \frac{V_E}{R_E} = \frac{V_{in}}{R_E}$$

The Emitter Current, $I_E$, is approximately equal to the Collector Current, $I_C$. The voltage across the collector resistor is:

$$V_{(\text{across } R_C)} = I_C R_C \approx I_E R_C$$

$$V_{(\text{across } R_C)} = V_{in} \frac{R_C}{R_E}$$

The Collector Gain factor, K, is thus

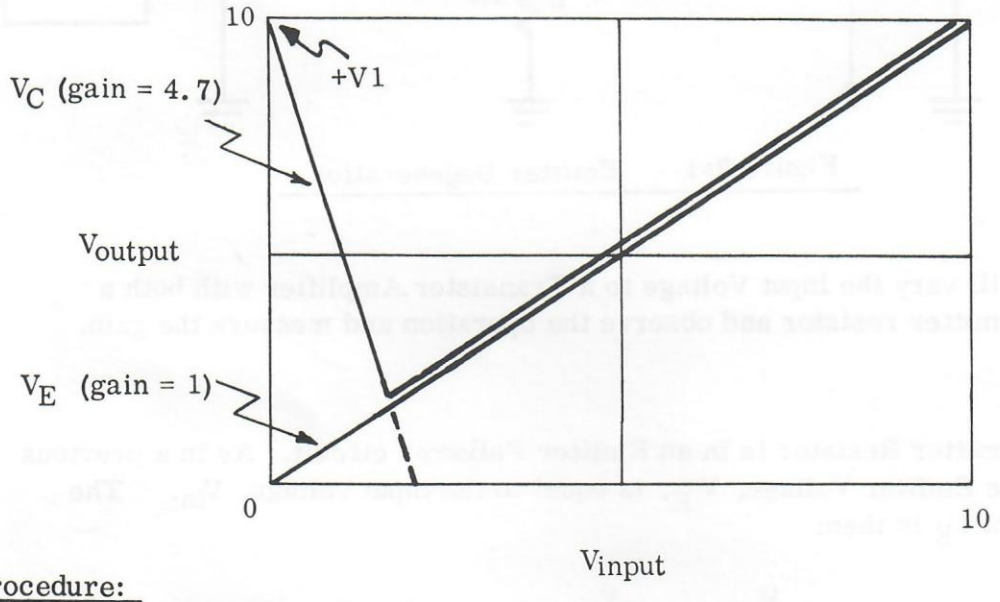
$$K = \frac{V_{(\text{ across } R_C)}}{V_{in}} = \frac{R_C}{R_E}$$

$$K = \frac{4.7K}{1K}$$

(COLLECTOR OUTPUT GAIN) $K = 4.7$

(Note: $V_C = +V1 - V_{(\text{across } R_C)}$

The expected output curves are given below:



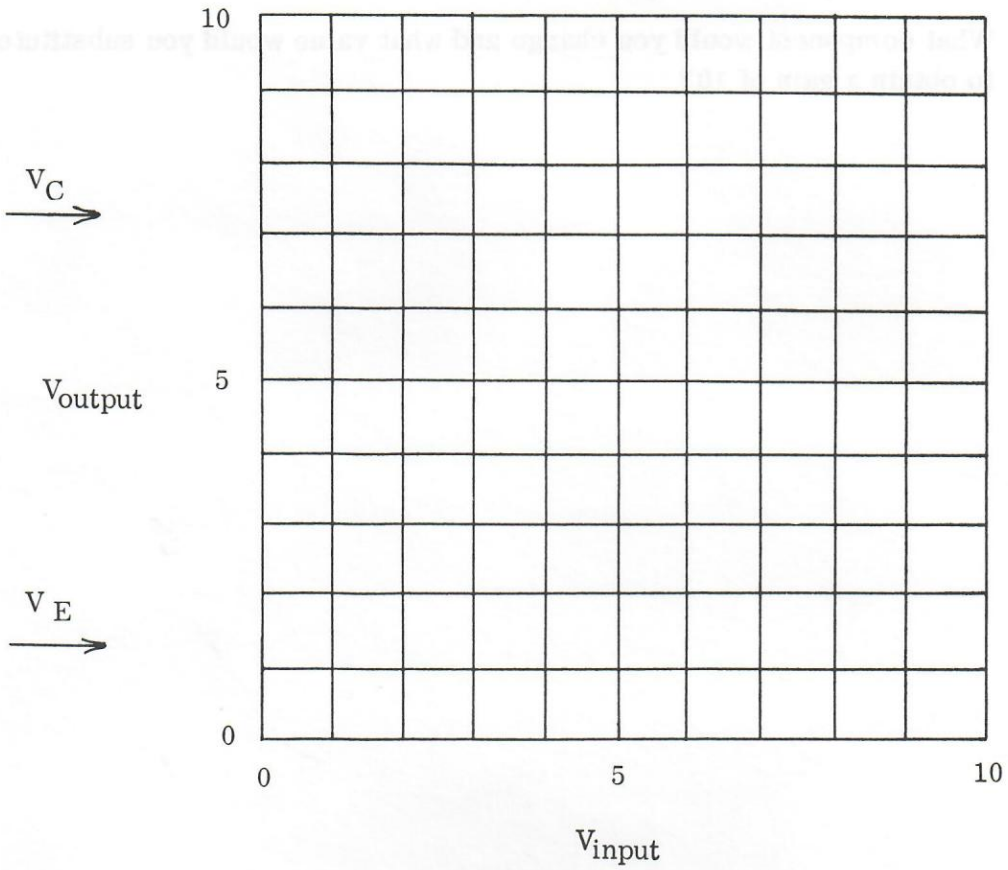

Procedure:

1. Wire the circuit shown in Figure 9 - 1.
2. Vary the Input Voltage (1K POT) and read the two Output Voltages $V_E$ and $V_C$. Enter the data in the table. Use the D.C. Meter on the Ed-Lab equipment.

| $V_{in}$ | $V_E$ | $V_C$ |
|---|---|---|
| 0 | | |
| .2 | | |
| .4 | | |
| .6 | | |
| .8 | | |
| 1 | | |

| $V_{in}$ | $V_E$ | $V_C$ |
|---|---|---|
| 1.5 | | |
| 2 | | |
| 4 | | |
| 6 | | |
| 8 | | |
| 9 | | |

3. Plot the results below:




4. Calculate the Gain by measuring the "slope" of the curves:

$$\text{COLLECTOR GAIN} = \frac{\Delta V_C}{\Delta V_{in}} = -$$

$$\text{EMITTER GAIN} = \frac{\Delta V_E}{\Delta V_{in}} = +$$

Do the Gains correspond to the theoretical values?

5. What is the linear operating range for the input voltage?

6. What is the linear operating range for the collector output voltage?

7. What BIAS point should you select?

8. What component would you change and what value would you substitute to obtain a gain of 10?

| Ed-Lab | EMITTER BY-PASS CAPACITOR | EXP. NO. 10<br>Page No. 1 |
|---|---|---|

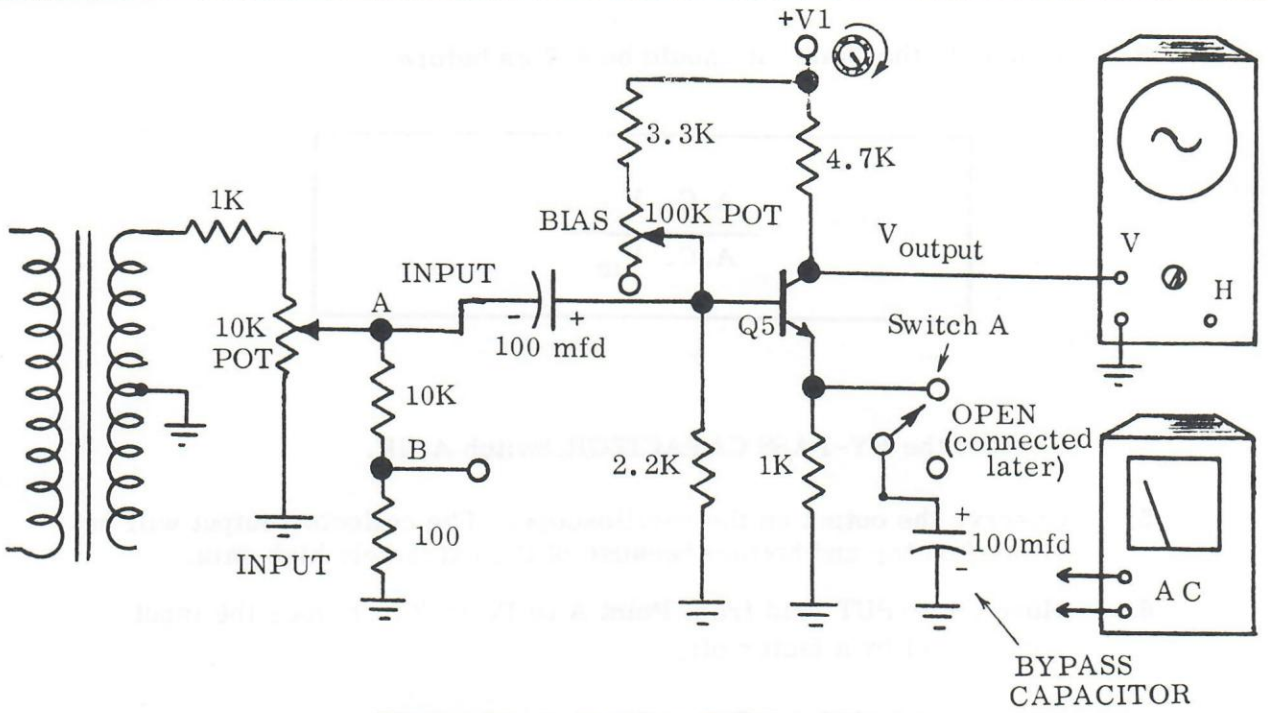



Figure 10 - 1 Emitter By-Pass Circuit

Objective:

You will use a CAPACITOR in parallel with the Emitter Resistor as an A.C. BYPASS. This does not affect the Bias Voltages but eliminates the Gain Degeneration. You will measure the Gain with and without the By-Pass Capacitor.

**Procedure:**

1. Wire the circuit in Figure 10 - 1.
   Do not connect the 100 MFD By-Pass Capacitor. Switch A DOWN.
   Connect the INPUT lead to Point A. This provides a large and variable input voltage.

2. Observe the output on the oscilloscope and adjust the Bias (100K POT) and Input level (10K POT) to yield an unclipped output. Use an external Oscilloscope to measure the input and output voltages.

| |
|---|
| A.C. $V_{in}$ (Point A) = |
| A.C. $V_{out}$ (Collector) = |

3. Calculate the Gain. It should be 4.7 as before.

$$\text{GAIN} = \frac{\text{A.C. } V_{out}}{\text{A.C. } V_{in}} =$$

4. Connect the BY-PASS CAPACITOR, Switch A UP.

5. Observe the output on the oscilloscope. The collector output will be clipped on top and bottom because of the extremely high gain.

6. Move the INPUT lead from Point A to Point B to reduce the input signal level by a factor of:

$$\frac{100}{10,100} \cong \frac{1}{100}$$

7. Adjust the INPUT level (10K POT) for a non-clipped linear output.

8. Measure the voltages at the output and at point A.

| |
|---|
| A.C. $V_{output}$ = |
| A.C. $V_{input}$ at point A = |

9. Calculate the Gain:

$$\text{A.C. } V_{input} \text{ at point B} = \frac{V_{input} \text{ at A}}{100} =$$

$$\text{GAIN} = \frac{\text{A.C. } V_{output}}{\text{A.C. } V_{input} \text{ at B}} =$$

| |
|---|
| GAIN = |

10. What is the function of the By-Pass Capacitor?

11. How does the high gain calculated above using the By-Pass Capacitor compare with the Common Emitter Amplifier Gain measured in Experiment No. 5?

**Ed-Lab** | DYNAMIC AMPLIFIER & PHASE SPLITTER | **EXP. NO. 11** **Page No. 1**

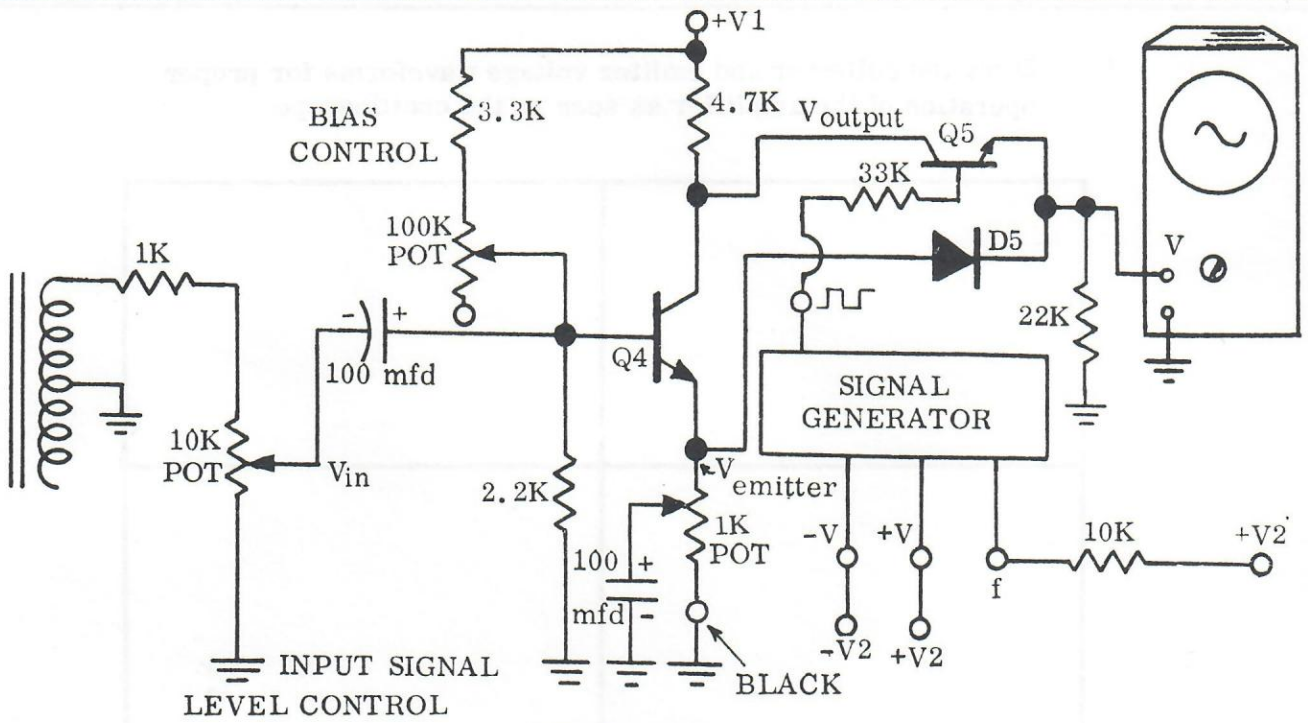



Figure 11-1

Objective:

You will apply an A.C. Input Signal to the Common Emitter Transistor Amplifier with an emitter bias resistor. Both the emitter bias voltage and the output collector voltage are viewed simultaneously on the oscilloscope with a diode and transistor used as a Dual Trace Oscilloscope Switching Device.

Procedure:

1. Wire the circuit in Figure 11-1. Set the 1K POT to 0 (NO BY-PASS).

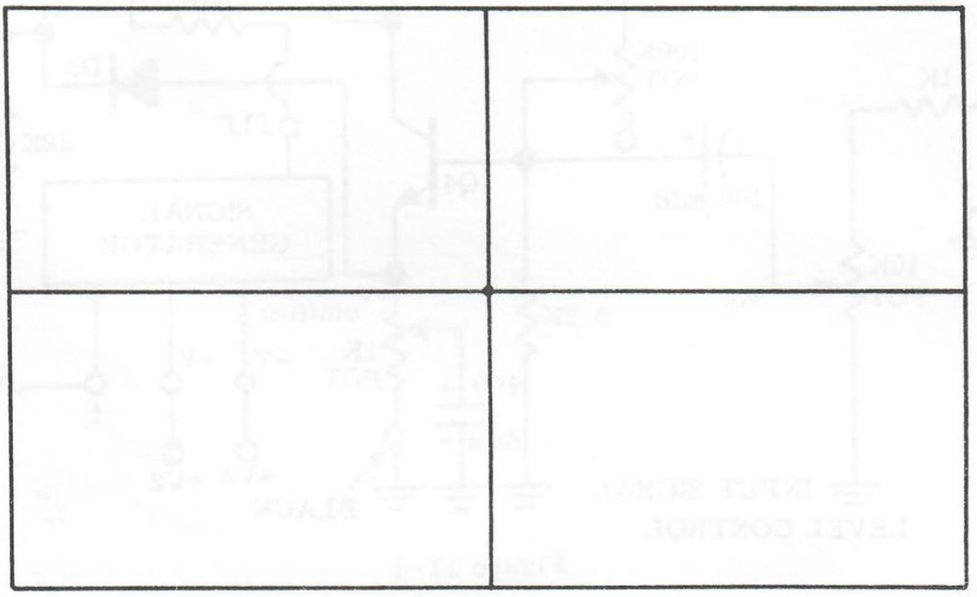
   This is the same amplifier used in the previous experiment with output dual-trace switching. The A.C. Input Signal is derived from the line transformer (60 Hertz).

   The 10K POT is a LEVEL CONTROL for the input signal voltage amplitude. The 1K POT provides for any amount of "by-pass".

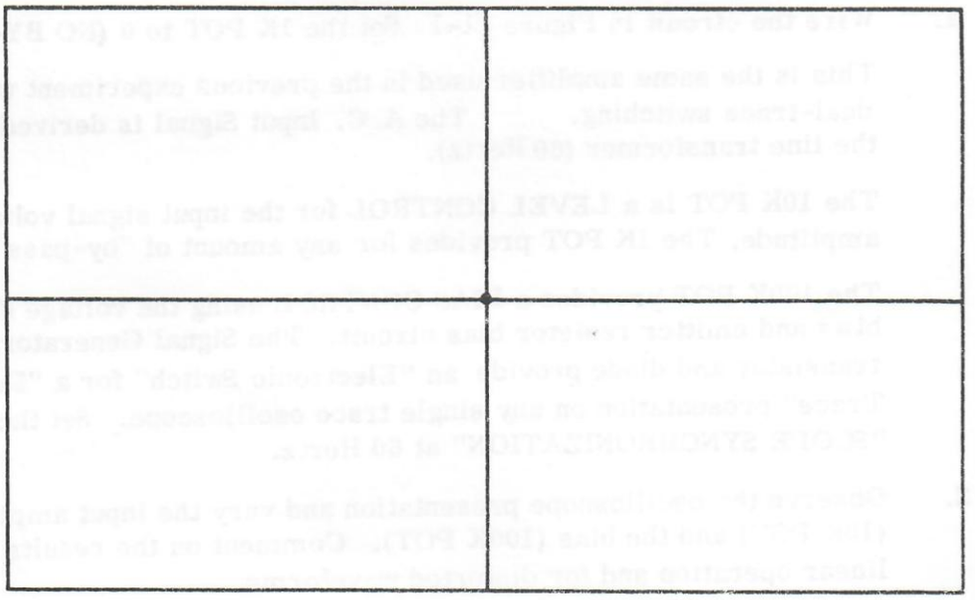
   The 100K POT provides a BIAS CONTROL using the voltage divider bias and emitter resistor bias circuit. The Signal Generator, the transistor and diode provide an "Electronic Switch" for a "Dual Trace" presentation on any single trace oscilloscope. Set the "SCOPE SYNCHRONIZATION" at 60 Hertz.

2. Observe the oscilloscope presentation and vary the input amplitude (10K POT) and the bias (100K POT). Comment on the results for linear operation and for distorted waveforms.

3. Draw the collector and emitter voltage waveforms for proper operation of the amplifier as seen on the oscilloscope.



4. Draw the waveform for a distorted output with an input signal which is too large. Adjust an input signal amplitude (10K POT) to result in clipping both on top and on bottom of the output waveform.

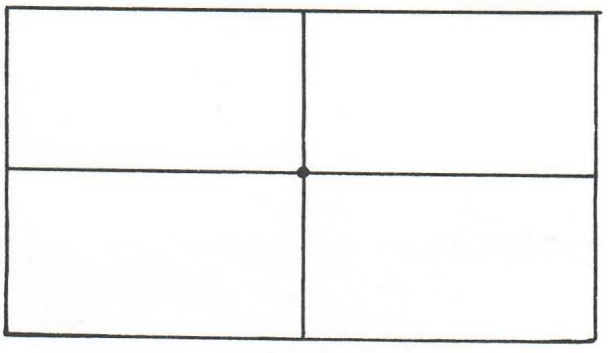


5. What is the maximum undistorted collector output A.C. signal voltage? What is the maximum permitted input A.C. voltage for an undistorted output?

6. What is the PHASE RELATIONSHIP between the collector output (C) and the emitter output (E)?

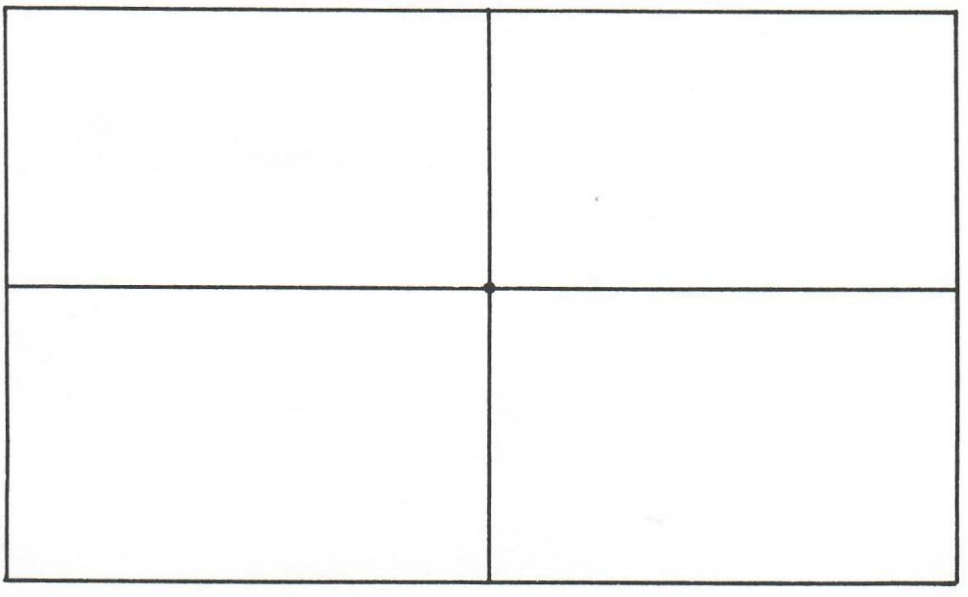
7. Vary the amount of "emitter by-pass" using the 1K Pot. What is the effect on the amplifier GAIN and BIAS ?

   Sketch the undistorted high gain output below:

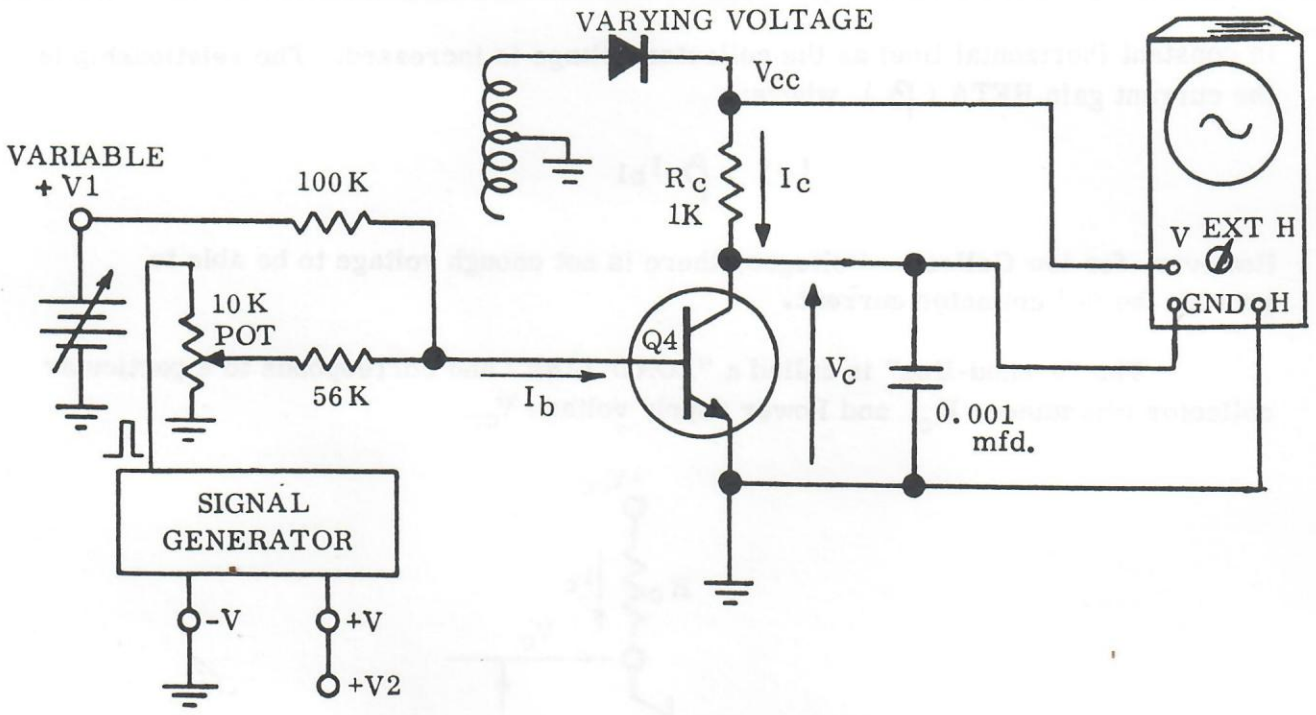


8. Substitute a 1K Resistor for the 4.7K collector resistor. The circuit is now a PHASE SPLITTER. Set the 1K POT to 0 to eliminate the By-Pass.

   Both the collector and emitter signals are of equal amplitude but in opposite phase. This is useful in driving "Push-Pull" Power Amplifiers which are described later. Substitute the 1K Resistor and draw the waveforms below:



| Ed-Lab | TRANSISTOR CURVE TRACER | EXP. NO. 12<br>Page No. 1 |
|---|---|---|

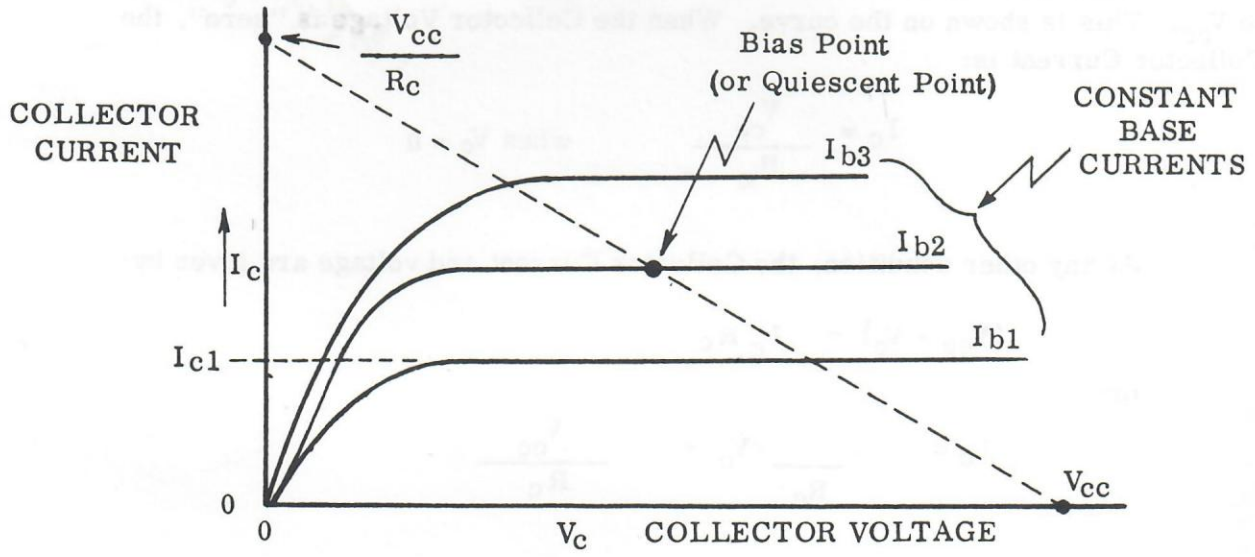



Objective:

You will construct a circuit which will automatically plot the Collector Characteristic Curve for the Transistor.

Description:

The Collector Characteristic Curves for a transistor is shown below:



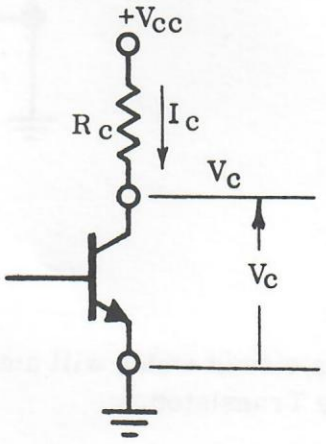

The graph is a plot of Collector Current, $I_c$, versus Collector Voltage, $V_c$, for constant Base Current, $I_b$. Different base currents are used to provide a "family of curves" as shown above. The curves show that the collector current

is constant (horizontal line) as the collector voltage is increased. The relationship is the current gain BETA ( $\beta$ ), where:

$$I_{c1} = \beta\ I_{b1}$$

However, for low Collector Voltages, there is not enough voltage to be able to provide the full collector current.

The "dashed-line" is called a "LOAD-LINE" and corresponds to a particular collector resistance, $R_c$, and Power Supply voltage $V_{cc}$.




When the collector current is "zero", the Collector Voltage, $V_c$, is equal to $V_{cc}$. This is shown on the curve. When the Collector Voltage is "zero", the Collector Current is:

$$I_c = \frac{V_{cc}}{R_c} \qquad \text{when } V_c = 0$$

At any other condition, the Collector Current and voltage are given by:

$$(V_{cc} - V_c) = I_c R_c$$

or:

$$I_c = -\frac{1}{R_c} V_c + \frac{V_{cc}}{R_c}$$

This is a straight line in the algebraic form of: $y = mx + b$ with an intercept and slope. The slope (m) of this line is $-\frac{1}{R_c}$ and the "y intercept" is $\frac{V_{cc}}{R_c}$. This line is shown as a "dashed line" on the "Characteristic Curves" and is the Load Line. With a fixed circuit, $V_{cc}$ and $R_c$, the "operating point" of the amplifier "slides" along this line. A typical "bias-point" is selected and sketched as a black dot. A larger collector resistor results in a lower slope.

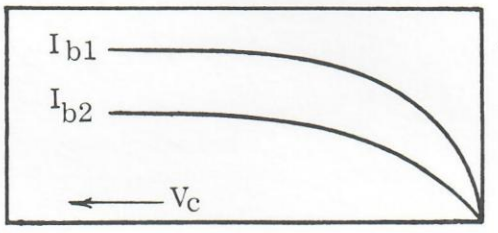
In designing a Common Emitter Transistor Amplifier, the Collector Characteristic Curves are used to determine the Power Supply Voltage, the Collector or Load Resistor, and the Bias Point.

Procedure:

1. Wire the circuit shown in Figure 12 - 1.
   Set the Oscilloscope to EXTERNAL HORIZONTAL input.
   The transformer 60 Hertz varying voltage automatically varies the collector voltage and also the collector current.
   The Oscilloscope Horizontal input is taken across the transistor ($V_c$) and the vertical input is the collector current, $I_c$, measured as the voltage across the collector resistor:

$$I_c = \frac{V_c}{R_c}$$

   The plot of the horizontal $V_c$ is actually reversed and the curve appears as shown below:




May appear reversed depending on the oscilloscope deflection direction.

   The signal generator "square wave" switches between two different base currents and thus two of the curves are plotted. Varying +V1 also varies the base current. Also vary the 10 K POT.

2. Plot the observed characteristic curves below:

| Ed-Lab | COMMON BASE AMPLIFIER | EXP. NO. 13<br>Page No. 1 |
|---|---|---|

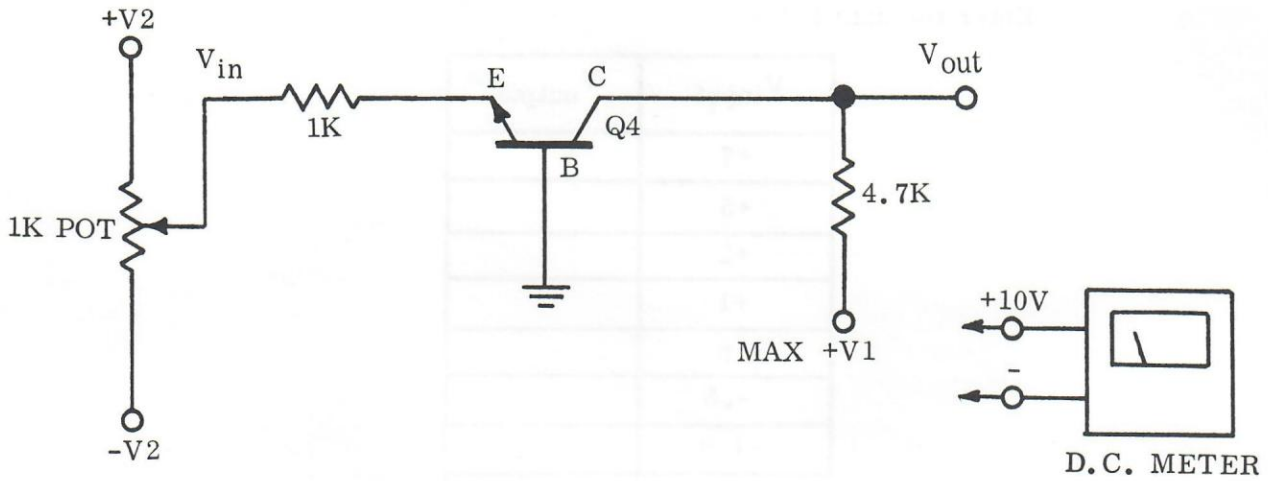



Figure 13-1 Common Base Amplifier

Objective:

You will apply an INPUT signal to the transistor EMITTER and observe the output from the COLLECTOR in a COMMON BASE transistor amplifier.

Description:

In the earlier discussions it was indicated that the current in the base-emitter circuit determines the current in the output collector circuit. Base-Emitter Current was controlled by applying a positive voltage to the Base thru a resistor. It is also possible to control the current by a negative INPUT voltage applied to the EMITTER. When the input is applied to the emitter and the output is obtained from the collector, the circuit is a Common Base Amplifier. This type of amplifier is not used very often but can operate at higher frequencies than in the Common Emitter Configuration and is used sometimes in very high frequency radio circuits. If an input resistor is in the base circuit, then the gain is the same as in the common emitter circuit. If the input resistor is in the emitter circuit (as in the circuit 1K) then the gain is the same as in the emitter degeneration circuit (Gain = 4.7).

Procedure:

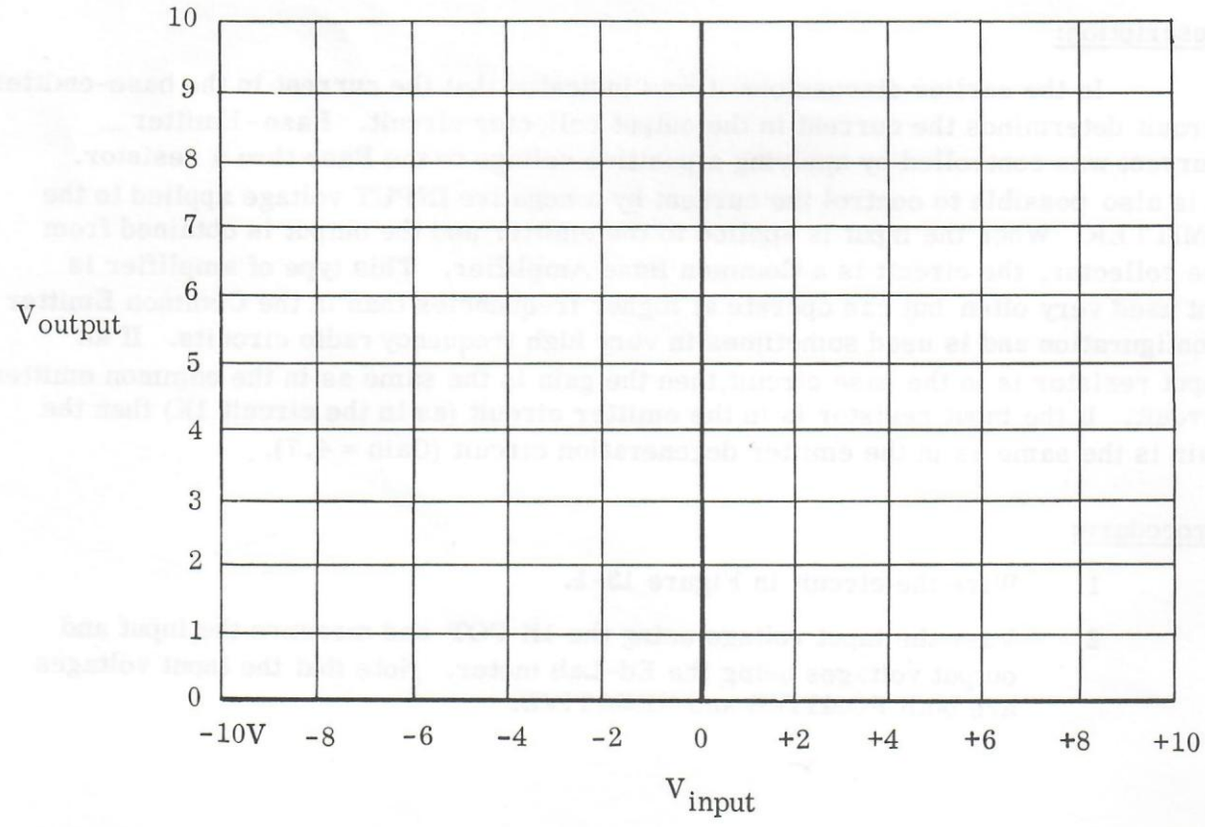
1. Wire the circuit in Figure 13-1.
2. Vary the input voltage using the 1K POT and measure the input and output voltages using the Ed-Lab meter. Note that the input voltages are both POSITIVE and NEGATIVE.

| Ed-Lab | COMMON BASE AMPLIFIER | EXP. NO. 13<br>Page No. 2 |
|---|---|---|

Enter the data below:

| $V_{input}$ | $V_{output}$ |
|---|---|
| +7 | |
| +5 | |
| +2 | |
| +1 | |
| 0 | |
| -.5 | |
| -1.0 | |
| -1.5 | |
| -2 | |
| -4 | |
| -6 | |
| -7 | |

3. Plot the results in the table below:



| Ed-Lab | COMMON BASE AMPLIFIER | EXP. NO. 13<br>Page No. 3 |
|---|---|---|

4. What is the Gain?

$$\text{Gain} = \frac{(\text{Change in } V_{out})}{(\text{Change in } V_{in})} =$$

| Gain = |
|---|

5. Is the Gain POSITIVE or NEGATIVE?

6. How does this Gain compare with the Gain in the Common Emitter Circuit in Experiment No. 9 with emitter degeneration?

7. At what input and output voltage would you select the mid operating bias point? Draw this point on the graph. Is the input bias positive or negative?

**Ed-Lab** | COMMON BASE A.C. AMPLIFIER | **EXP. NO.** 14 **Page No.** 1

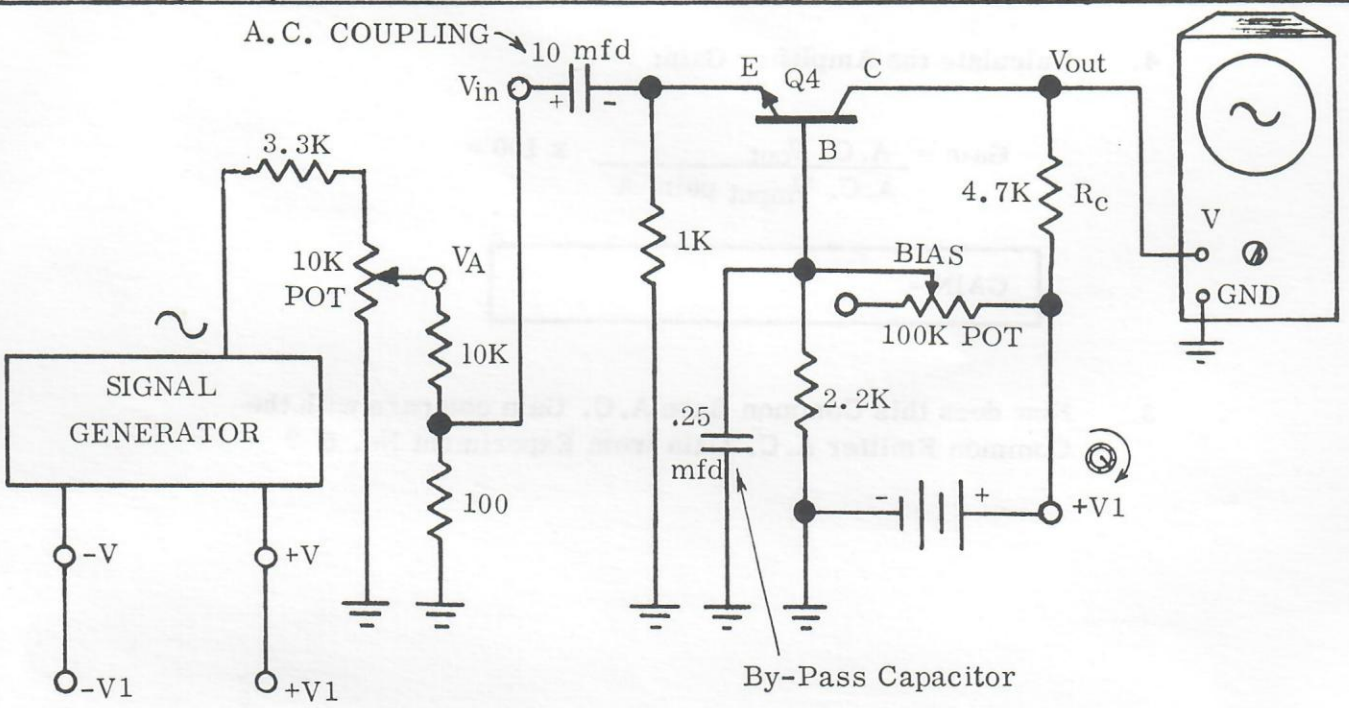



Figure 14 - 1 Common Base A.C. Amplifier

Objective:

You will use a Common Base Amplifier with a VOLTAGE DIVIDER BIAS circuit and a BY-PASS CAPACITOR and measure the amplifier GAIN.

Description:

Various techniques can be used to apply a BIAS for the Common Base Amplifier. Either a positive voltage bias is required on the base or a negative voltage bias is required on the emitter. A voltage divider circuit provides a positive bias on the Base, and a By-Pass Capacitor is used to maintain the high gain by eliminating "degeneration" as in previous amplifiers. The SIGNAL GENERATOR sine wave is used.

Procedure:

1. Wire the circuit in Figure 14-1.
2. Adjust the BIAS (100K POT) and the INPUT Level (10K POT) for a non-distorted output.
3. Use the Oscilloscope to measure $V_{out}$ and the input voltage at $V_A$. Enter the data below:

| |
|---|
| A. C. $V_{out}$ = |
| A. C. $V_{in}$ at point A = |

4. Calculate the Amplifier Gain:

$$\text{Gain} = \frac{\text{A.C. } V_{out}}{\text{A.C. } V_{input} \text{ point A}} \times 100 =$$

| GAIN = |
|---|

5. How does this Common Base A.C. Gain compare with the Common Emitter A.C. Gain from Experiment No. 5 ?

**Ed-Lab** | CASCADED AMPLIFIER | **EXP. NO.** 15 **Page No.** 1

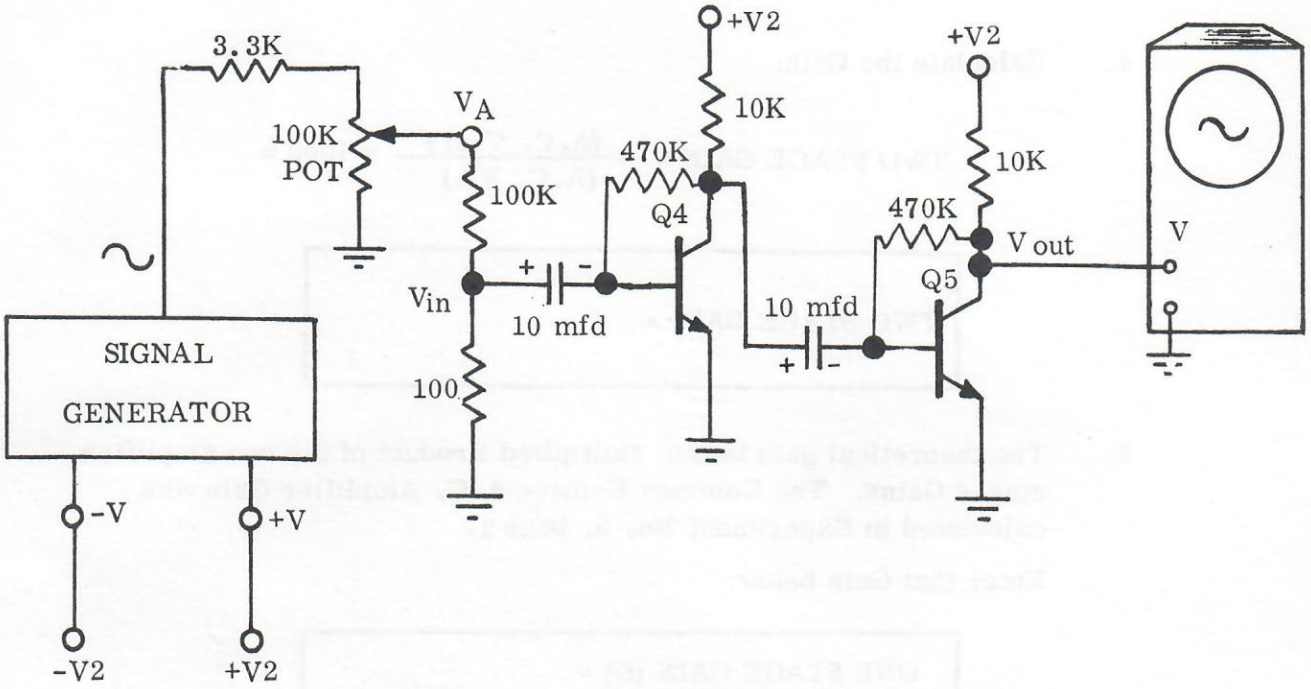



Figure 15 - 1 Two Stage Cascaded Amplifier

Objective:

You will connect two transistor amplifiers in "series" or " CASCADE" with CAPACITOR COUPLING to yield a very high gain amplifier for very low level input signals.

Procedure:

1. Wire the circuit in Figure 15 - 1.
2. Monitor the output on the oscilloscope and adjust the input (100K POT) to yield an undistorted output sine wave.
3. Read the A.C. voltages at the output and at point A. Note that:

$$V_{in} = V_A \cdot \frac{100}{100K}$$

$$V_{in} = \frac{V_A}{1000}$$

| |
|---|
| A.C. $V_{out}$ = |
| A.C. $V_A$ = |

4. Calculate the Gain:

$$\text{TWO STAGE GAIN} = \frac{(\text{A.C. Vout})}{(\text{A.C. V A})} \times 1000 =$$

| TWO STAGE GAIN = |
|---|

5. The theoretical gain is the Multiplied Product of the two amplifier stages Gains. The Common Emitter A.C. Amplifier Gain was calculated in Experiment No. 5, page 2.

Enter that Gain below:

| ONE STAGE GAIN (G) =<br>(Experiment No. 5 ) |
|---|

Theoretical two-stage gain is

$$\text{THEORETICAL TWO STAGE} = G \cdot G$$

| THEORETICAL GAIN = |
|---|

How do the theoretical and measured two stage gains compare?

6. Calculate the actual input voltage.

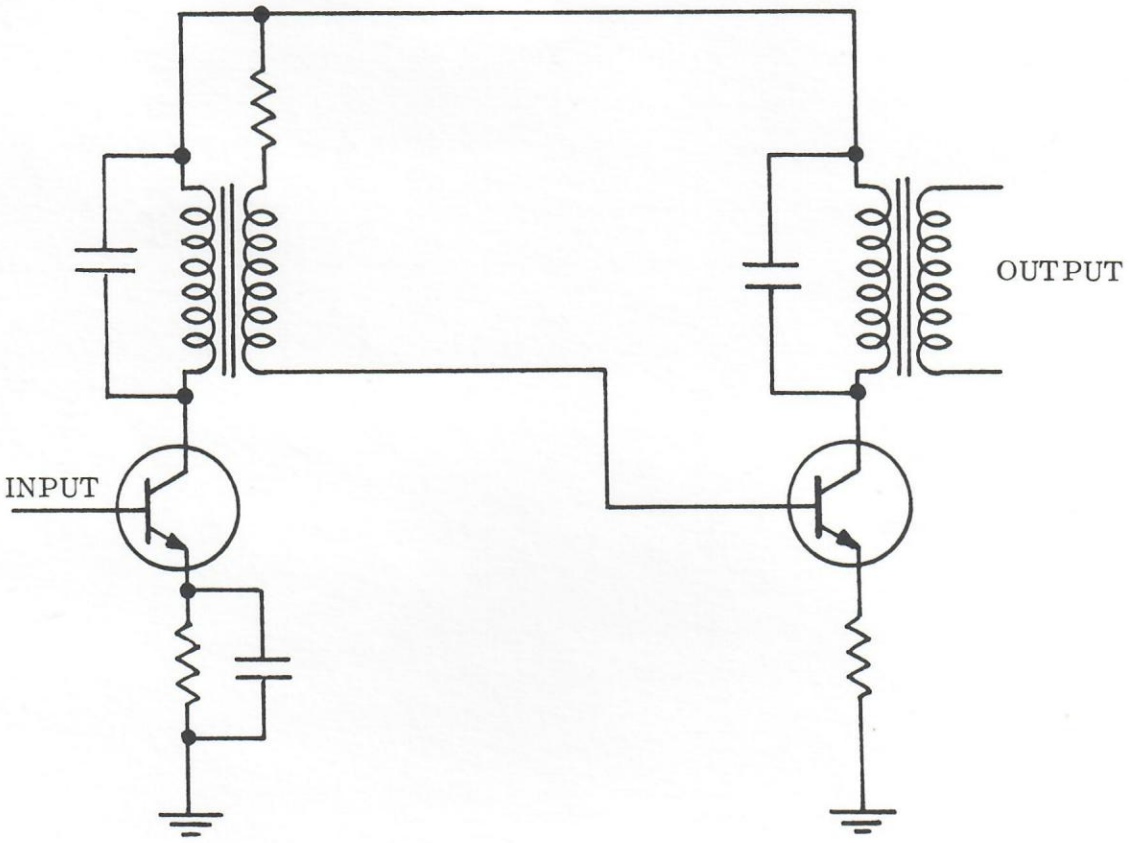
$$\text{A.C. } V_{in} = \frac{\text{A.C. VA}}{1000}$$

| A.C. $V_{in}$ = |
|---|

7. What is the Gain for a THREE STAGE CASCADED AMPLIFIER?

Transformer Coupling

A Cascaded Amplifier can also be "coupled" using a "transformer". The transformer can act as an A.C. coupling and can also use a capacitance to select certain frequencies as well as matching impedance output to impedance input. A typical transformer coupled stage is shown below:



| Ed-Lab | PNP TRANSISTOR | EXP. NO. 16<br>Page No. 1 |
|---|---|---|

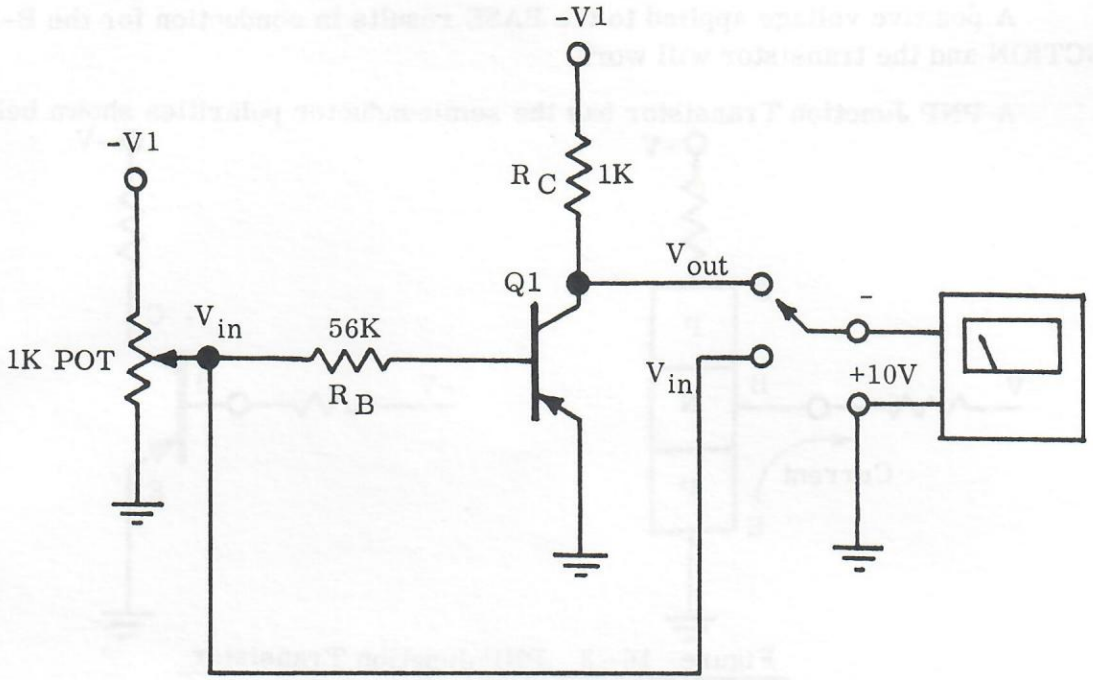



Figure 16 - 1 PNP Amplifier

Objective:

You will use a PNP-Transistor in a D.C. Amplifier, measure the Amplifier Gain and calculate the Current Gain or Beta.

Description:

The NPN "Junction" transistor consists of three Semiconductor "polarities" of negative-positive-negative as shown below:

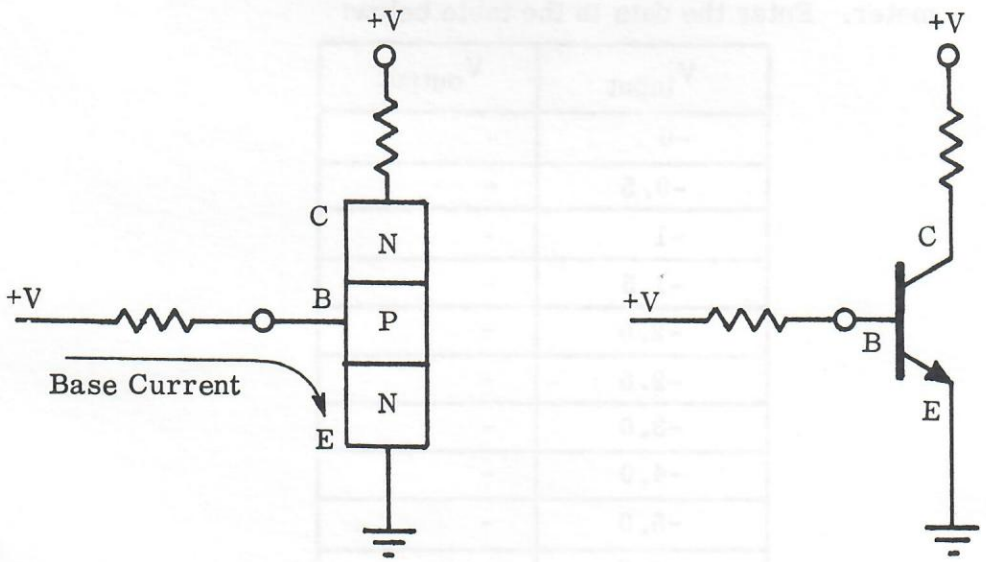



Figure 16 - 2 NPN Junction Transistor

A positive voltage applied to the BASE results in conduction for the B-E JUNCTION and the transistor will work.

A PNP Junction Transistor has the semiconductor polarities shown below:

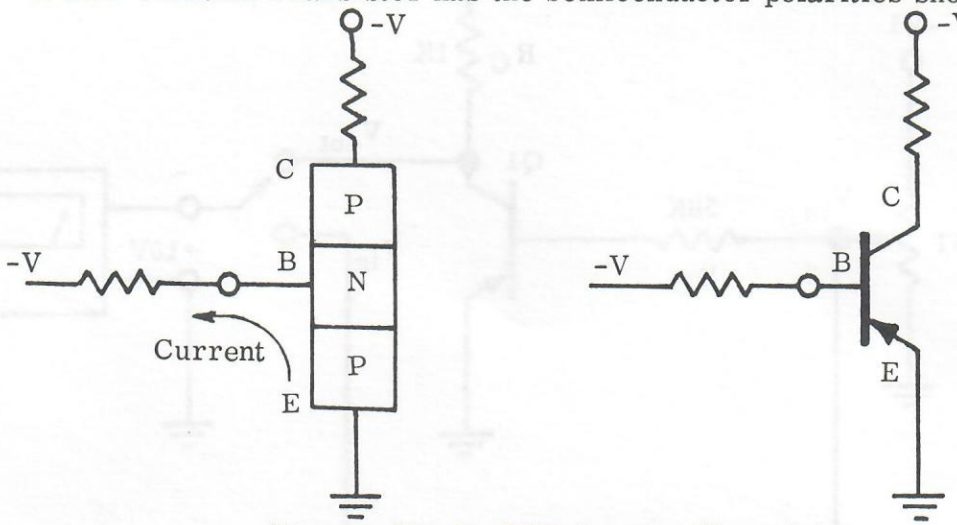



Figure 16-3 PNP Junction Transistor

A negative voltage must be applied to the BASE for conduction in the Base-Emitter Junction. A NEGATIVE COLLECTOR voltage must then also be used. The difference between the PNP and the NPN is that all Positive Voltages used in the NPN circuits must be changed to Negative Voltages in the PNP circuits. Otherwise, all the circuits studied remain the same.

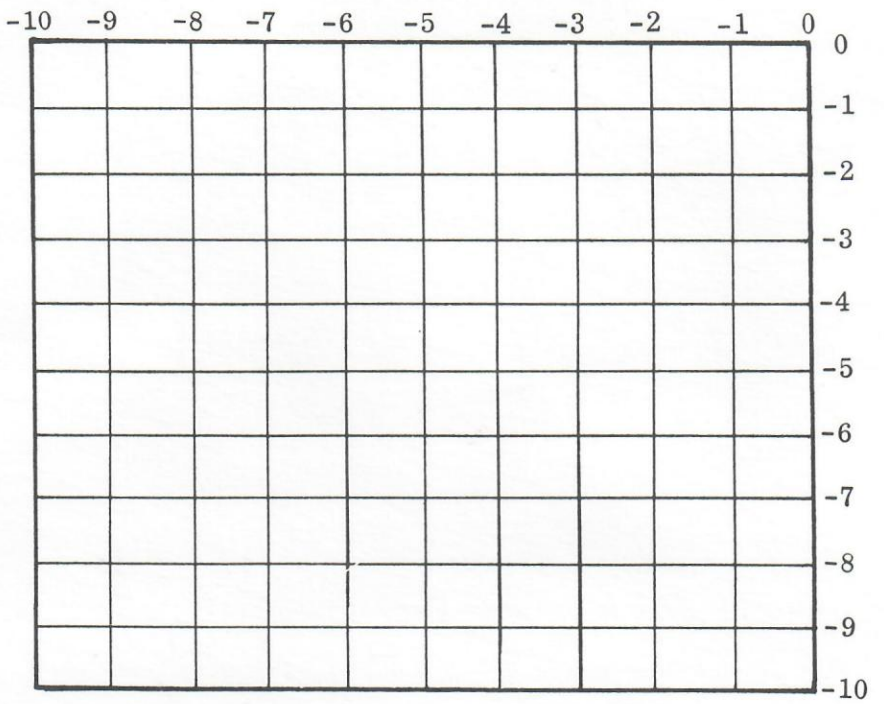
Procedure:

1. Wire the circuit in Figure 16-1 .
2. Vary the input voltage and read both input and output voltages on the meter. Enter the data in the table below:

| $V_{input}$ | $V_{output}$ |
|---|---|
| -0 | - |
| -0.5 | - |
| -1 | - |
| -1.5 | - |
| -2.0 | - |
| -2.5 | - |
| -3.0 | - |
| -4.0 | - |
| -5.0 | - |
| -6.0 | - |
| -8.0 | - |

| Ed-Lab | PNP TRANSISTOR | EXP. NO. 16<br>Page No. 3 |
|---|---|---|

3. Plot the results below:




4. Calculate the Amplifier Voltage Gain:

$$\text{Gain} = \frac{\triangle V_{out}}{\triangle V_{in}} = -$$

| Gain (G) = — |
|---|

Is the Gain Positive or Negative?

5. Calculate the Current Gain, Beta, ( $\beta$ ):

$$G = \beta \cdot \frac{R_C}{R_B}$$

$$\text{or } \beta = G \cdot \frac{R_B}{R_C}$$

$$\beta = G \cdot \frac{56K}{1K}$$

$$\beta = 56 \cdot G$$

| PNP TRANSISTOR $\beta$ = |
|---|

**Ed-Lab** | NPN - PNP CASCADED AMP | **EXP. NO.** 17 **Page No.** 1

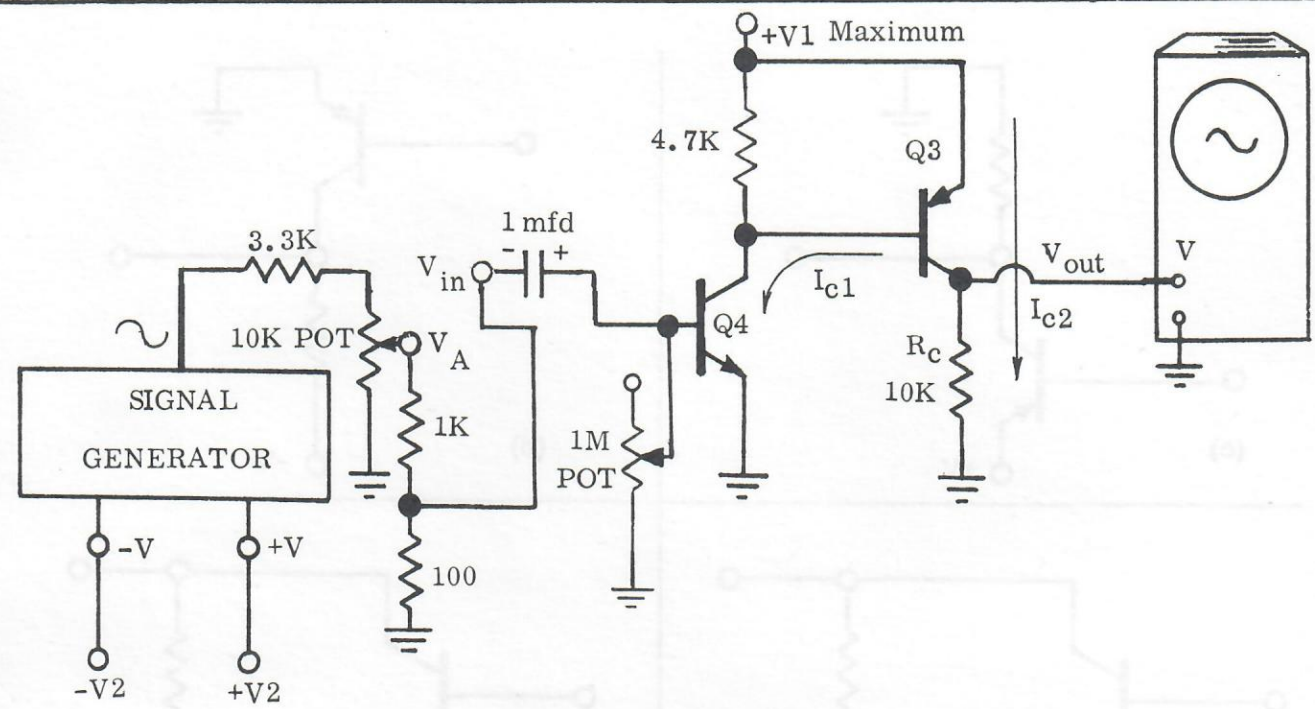



Figure 17 - 1 Complementary Cascade Amplifier

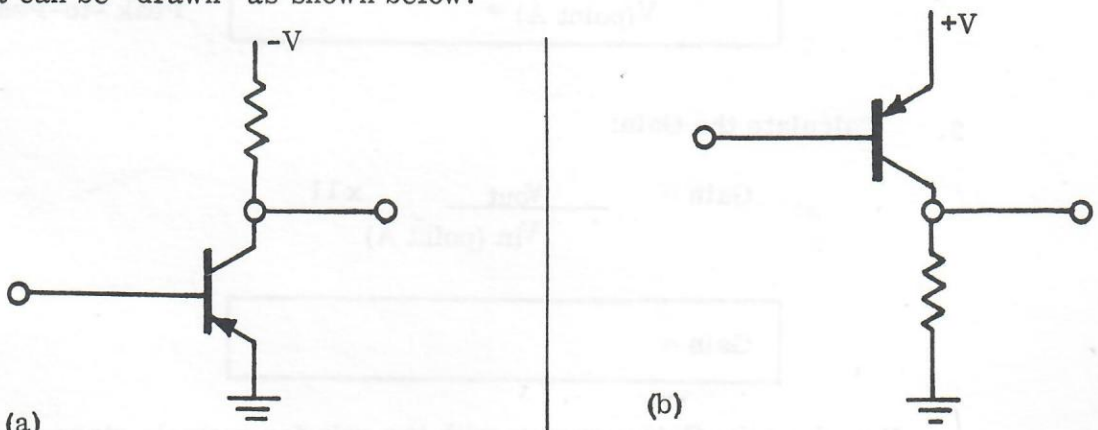
Objective:

You will connect two Common Emitter Amplifiers in CASCADE using an NPN stage and a PNP stage.

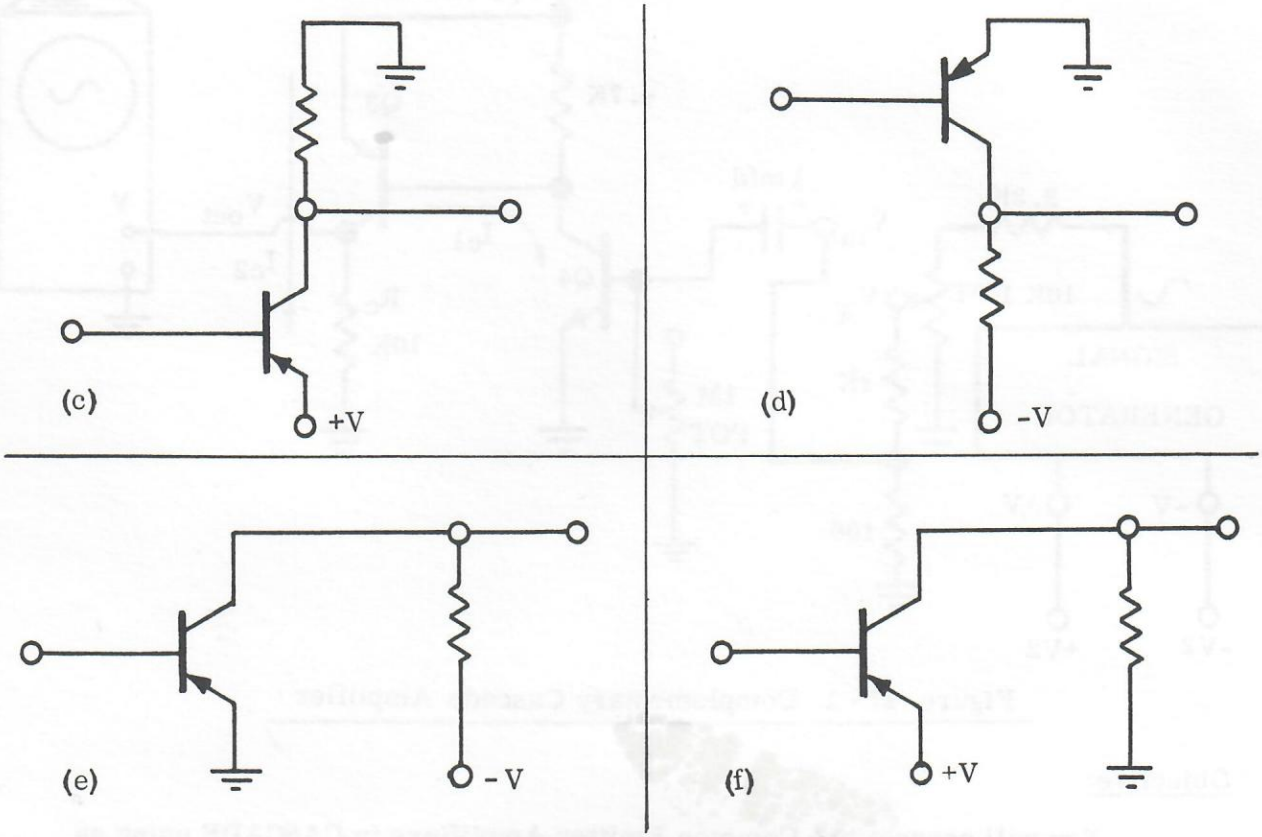
Description:

The collector current of the first stage NPN transistor is the base current of the second stage PNP transistor. The current gain of the pair is thus the product of BETA (Q4) x BETA (Q1).

Although a negative voltage was used in the previous experiment, the only requirement in a PNP transistor is that the BASE be more negative than the Emitter. There are many ways in which the SAME COMMON EMITTER AMPLIFIER circuit can be "drawn" as shown below:



| Ed-Lab | NPN - PNP CASCADED AMP | EXP. NO. 17<br>Page No. 2 |
|---|---|---|




Procedure:

1. Wire the circuit in Figure 17 - 1. Adjust the input amplitude (10K POT) and the 1M POT for an undistorted output.
2. Measure the Voltages to determine the Gain:

| | |
|---|---|
| V out = | Peak-to-Peak A.C. |
| V(point A) = | Peak -to-Peak A.C. |

3. Calculate the Gain:

$$\text{Gain} = \frac{V_{out}}{V_{in}\ (\text{point A})} \times 11$$

| |
|---|
| Gain = |

4. How does the Gain compare with the gain for a single stage amplifier?

PNP - NPN Combination:

PNP and NPN transistors can be connected in various different amplifier combinations as shown below:

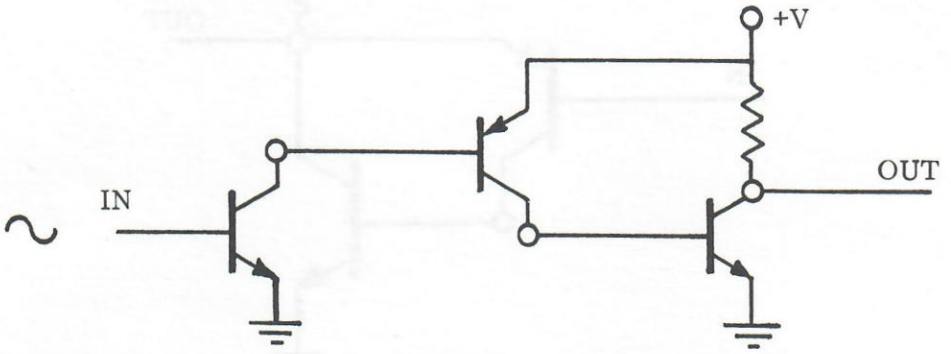



Figure 17-2 Three Stage NPN - PNP - NPN Amplifier

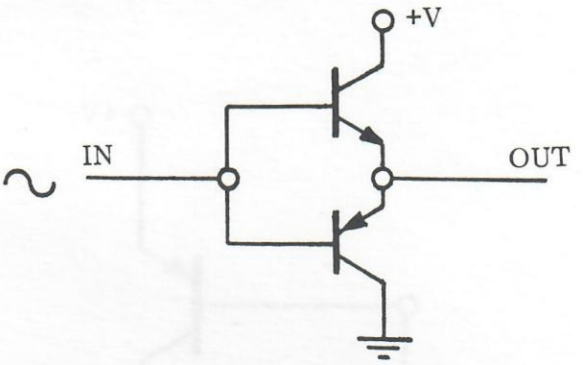



Figure 17-3 Complementary Emitter Follower "Driver"

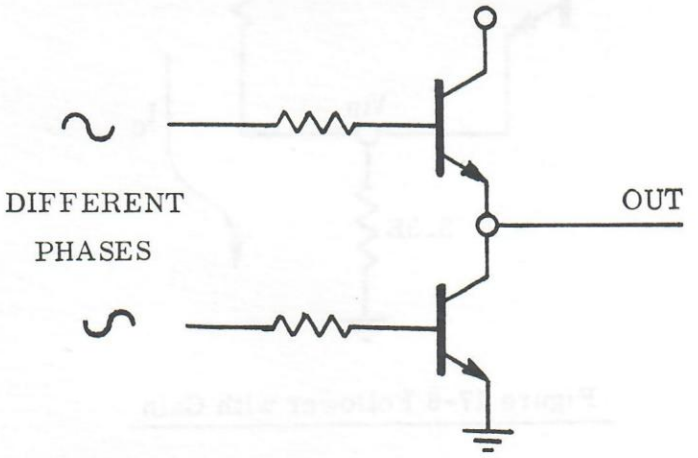



Figure 17 - 4 "Cascade" or "Totem Pole" Output Amplifier

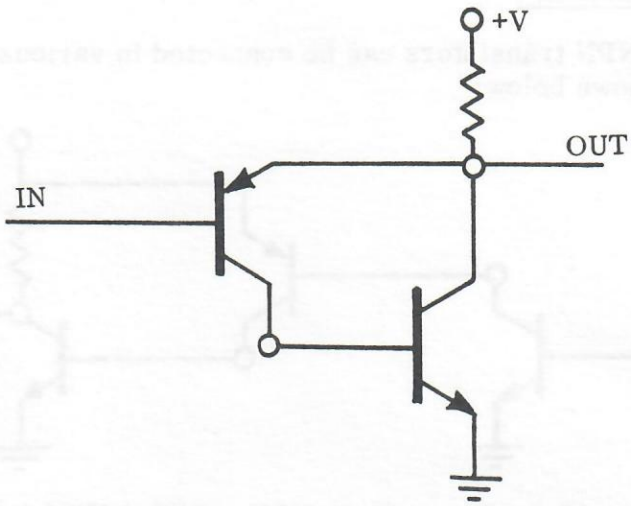
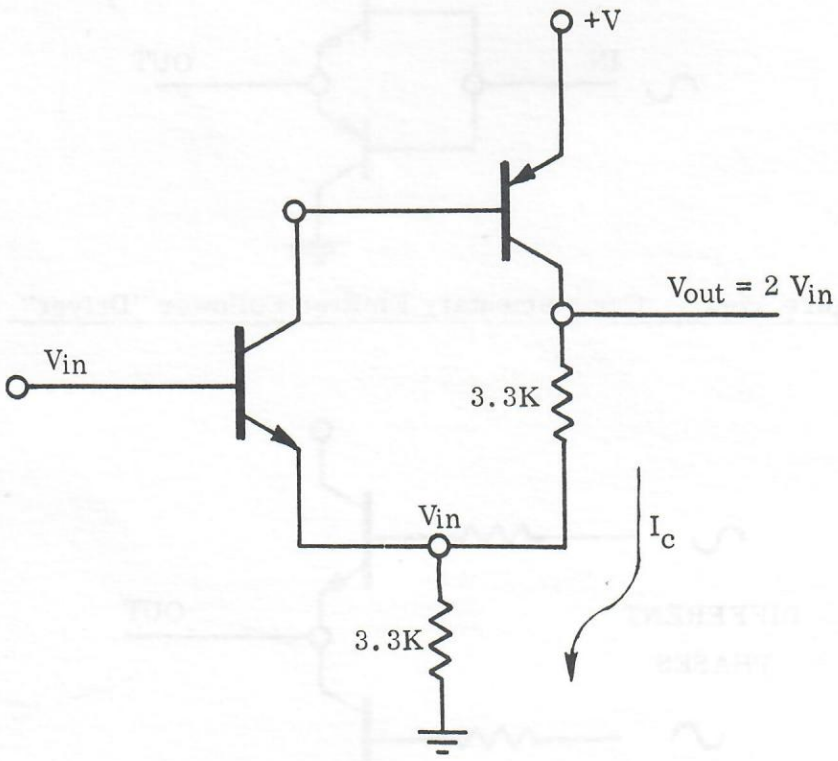



Figure 17 - 5 Pre-Driver Stage




GAIN = 2

Figure 17-6 Follower with Gain

| Ed-Lab | CONSTANT CURRENT SOURCE | EXP. NO. 18<br>Page No. 1 |
|---|---|---|

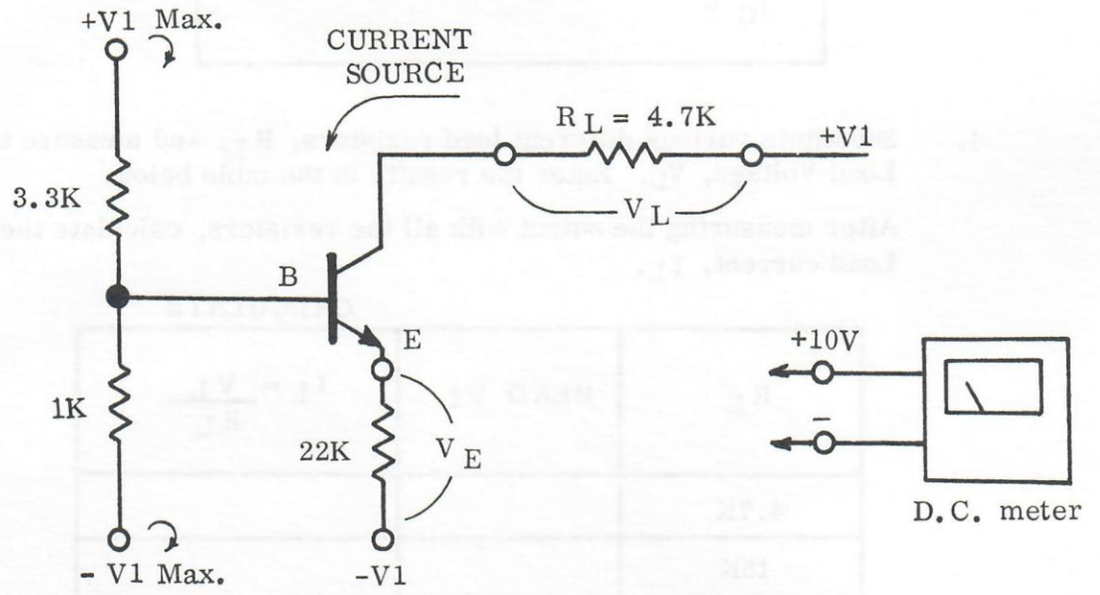



Figure 18-1 Constant Current Source

Objective:

You will use an Emitter Follower Circuit with a Collector Output to yield a Constant Current Source. You will determine the current for various Load resistors.

Description:

The Base Voltage is determined by the voltage divider (4.7K and 1K) and the Emitter Voltage is equal to the Base Voltage. The Emitter Current is equal to the Collector or Output Current and is:

$$I_C = I_E = \frac{V_E}{22K}$$

Procedure:

1. Wire the Circuit in Figure 18 - 1.
2. Measure the Voltage across the Emitter Resistor, $V_E$, to determine the current.

| $V_E$ = |
|---|

3. Calculate the Constant Current:

$$I_C = \frac{V_E}{22K} =$$

$I_C$ =

4. Substitute various different load resistors, $R_L$, and measure the Load Voltage, $V_L$. Enter the results in the table below.

   After measuring the output with all the resistors, calculate the Load current, $I_L$.

| $R_L$ | READ $V_L$ | CALCULATE $I_L = \frac{V_L}{R_L}$ |
|---|---|---|
| 4.7K | | |
| 15K | | |
| 33K | | |
| 56K | | |
| 100K | | |
| 470K | | |
| 1M | | |
| 10K | | |
| 3.3K | | |
| 2.2K | | |
| 1K | | |
| 100 | | |
| 0 Short | | |

5. Is the Load Current constant?

6. Why is the load current reduced for high resistance values?

| Ed-Lab | DIFFERENCE CIRCUIT | EXP. NO. 19<br>Page No. 1 |
|---|---|---|

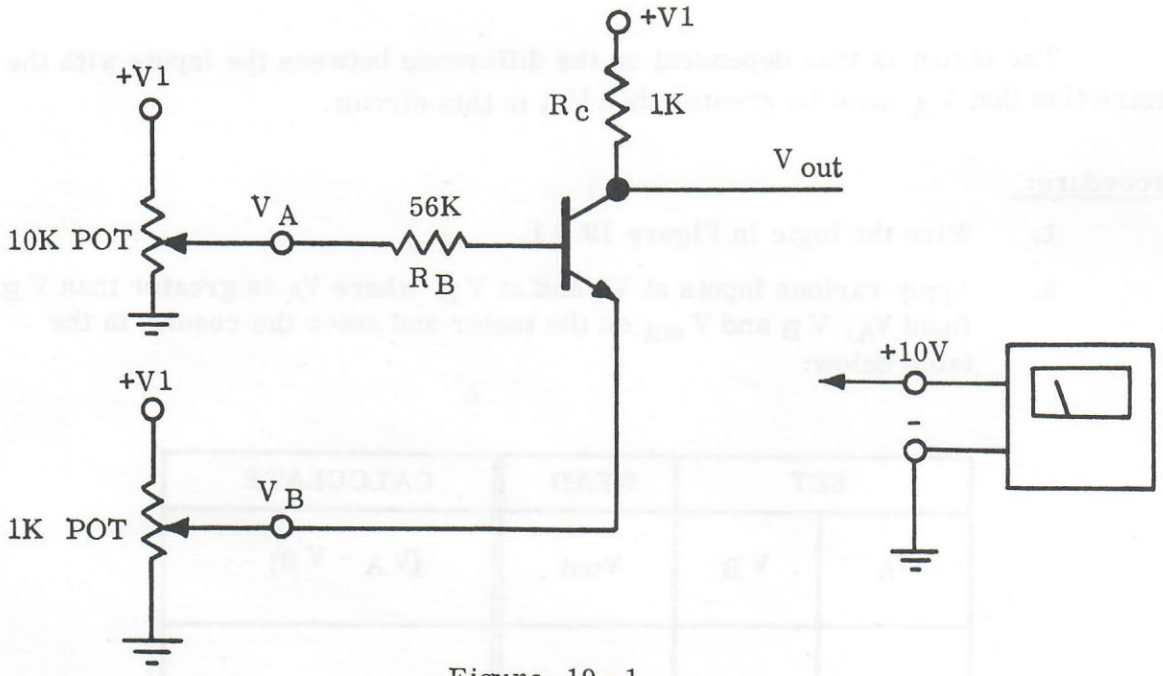



Figure 19-1

Objective:

You will use a Transistor Amplifier to provide an output which is equal to the difference between two inputs.

Description:

As described in earlier experiments, the Collector current was determined by BETA times the Base Current and the Base Current was determined by the Voltages at the Base Input and at the Emitter and by the series Base Resistor. In the Common Emitter Amplifier, the input was applied to the Base. In the Common Base Amplifier, the input was applied to the Emitter. If two inputs are applied to the Base and Emitter, the result is an INPUT DIFFERENCE CIRCUIT. The Base Current is:

$$I_B = \frac{(V_A - V_B)}{R_B}$$

The Collector Current is ($I_B$ x BETA).

The Output Voltage is:

$$V_{out} = I_C \cdot R_C = I_B \cdot \beta \cdot R_C$$

$$\boxed{V_{out} = (V_A - V_B)\ \beta\ \frac{R_C}{R_B}}$$

The output is thus dependent on the difference between the inputs with the restriction that $V_A$ must be greater than $V_B$ in this circuit.

Procedure:

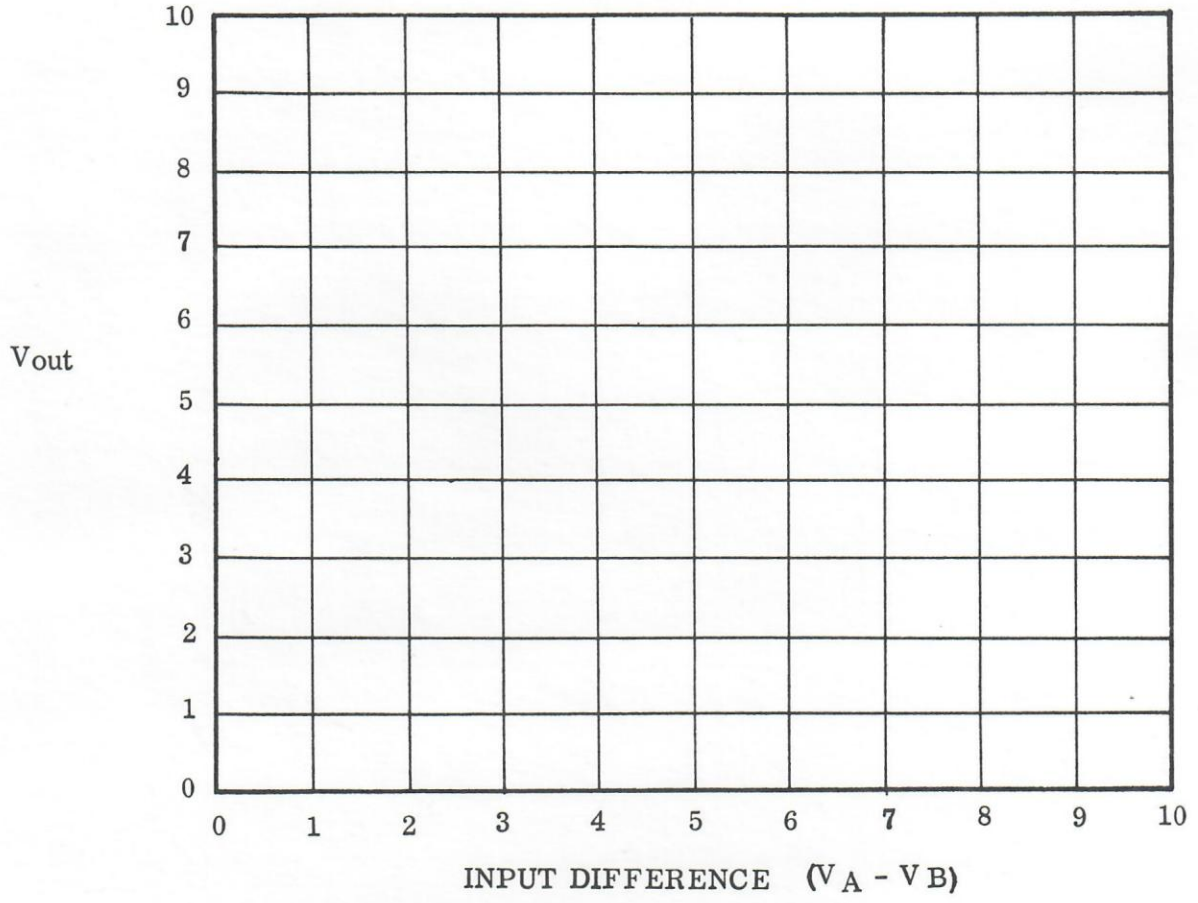
1. Wire the logic in Figure 19 - 1.
2. Apply various inputs at $V_A$ and at $V_B$ where $V_A$ is greater than $V_B$. Read $V_A$, $V_B$ and $V_{out}$ on the meter and enter the results in the table below:

| SET | | READ | CALCULATE |
|---|---|---|---|
| $V_A$ | $V_B$ | Vout | $(V_A - V_B)$ |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |

| Ed-Lab | DIFFERENCE CIRCUIT | EXP. NO. 19<br>Page No. 3 |
|---|---|---|

3. Calculate the difference ($V_A - V_B$) and enter the results in the table.

Plot the results below:




4. Does the output correspond to the difference between inputs?

5. What is the Difference Gain?

$$\text{DIFFERENCE GAIN} = \text{slope} = \frac{\Delta V_{out}}{\Delta(V_A - V_B)} =$$

| DIFFERENCE GAIN = |
|---|

6. Is the Difference Gain positive or negative?

| Ed-Lab | DIFFERENCE AMPLIFIER | EXP. NO. 20<br>Page No. 1 |
|---|---|---|

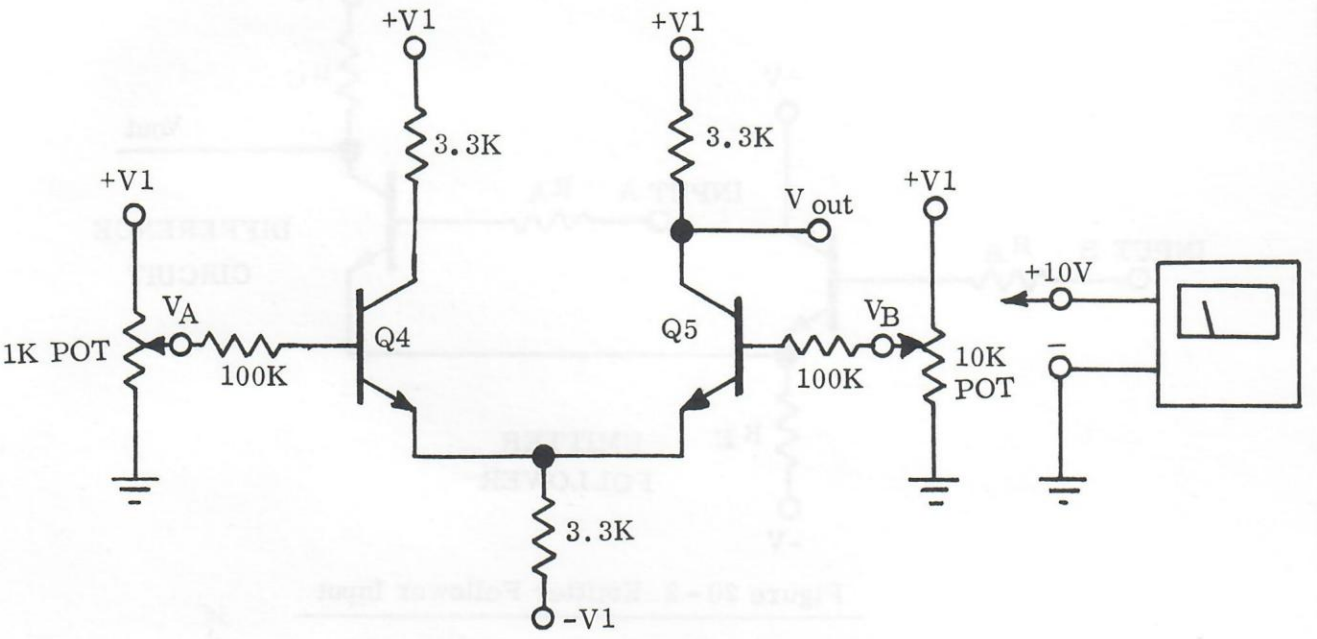



Figure 20 - 1 Difference Amplifier

Objective:

You will use two transistors to provide a DIFFERENCE or DIFFERENTIAL AMPLIFIER where either of the inputs can be larger.

Description:

The difference circuit in Exp. 19 had two basic restrictions: 1) The Base Input Voltage was required to be greater than the Emitter Input Voltage, and 2) The Emitter Input was required to be available from a Low Impedance Source, otherwise, there would be a Source Emitter Resistance and Emitter Degeneration and reduction in gain would occur and be dependent on the emitter source resistance. The idea is then to use an Emitter Follower on the Emitter input to provide a high impedance to the input source, and a low impedance to the Difference Circuit Emitter Input. The new circuit is shown on the following page:

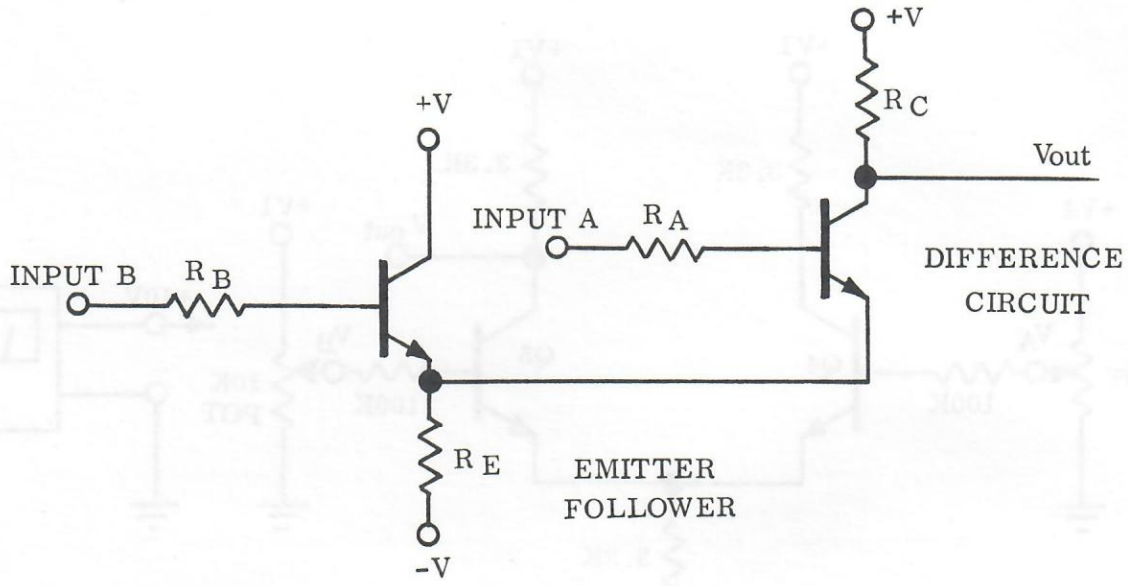



Figure 20 - 2 Emitter Follower Input

This same circuit can be redrawn as:

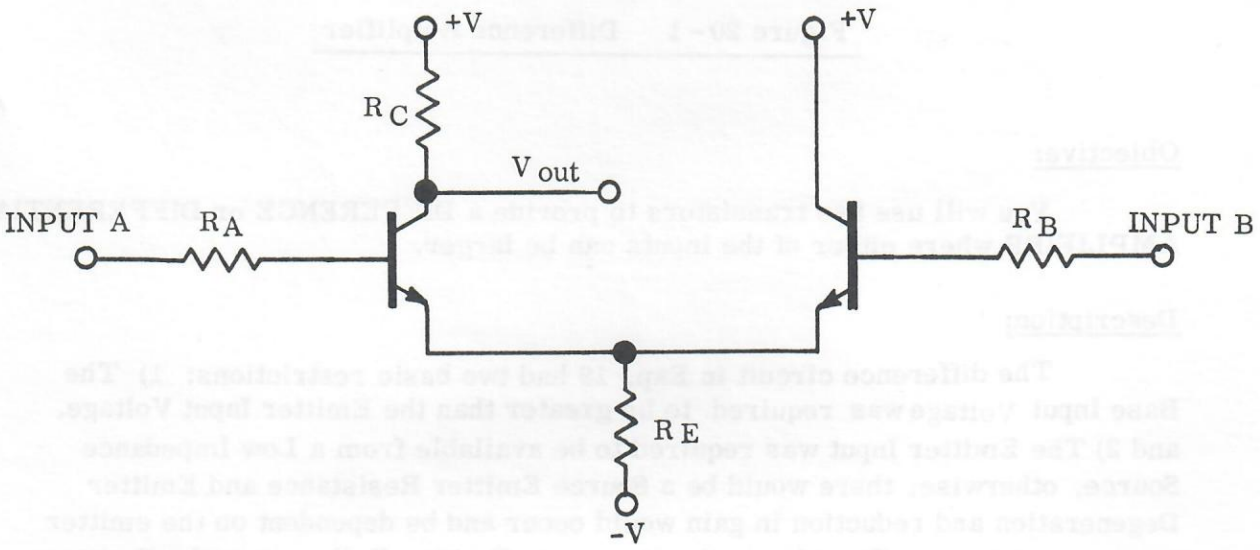



Figure 20 - 3 Difference Amplifier

This circuit is called a difference amplifier, and is "symmetrical" if we add a collector resistor to the B Input Transistor as in Figure 20 - 1. This circuit will operate with either a larger A or B input.

| Ed-Lab | DIFFERENCE AMPLIFIER | EXP. NO. 20<br>Page No. 3 |
|---|---|---|

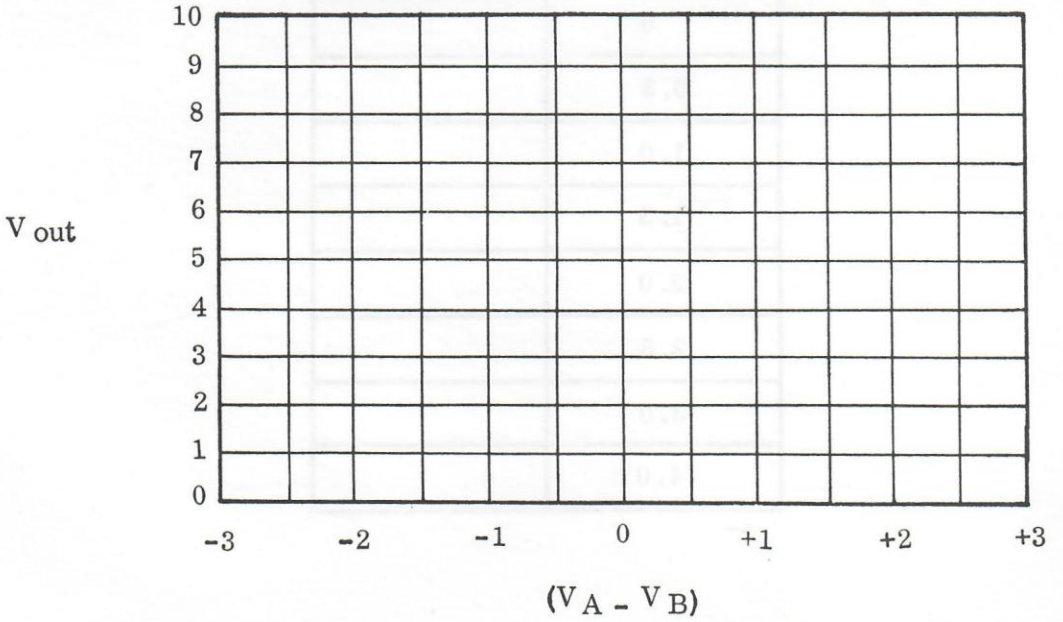
Procedure:

1. Wire the Difference Amplifier Circuit in Figure 20 - 1.

2. Vary inputs A and B and enter the data in the table below:

| $V_A$ | $V_B$ | $V_{out}$ | $(V_A - V_B)$ |
|---|---|---|---|
| 0 | 0 | | 0 |
| 0 | .5 | | -0.5 |
| 0 | 1 | | -1.0 |
| 0 | 1.5 | | -1.5 |
| 0 | 2 | | -2.0 |
| .5 | 0 | | +.5 |
| 1.0 | 0 | | +1.0 |
| 1.5 | 0 | | +1.5 |
| 2 | 0 | | +2 |
| .5 | 1 | | -.5 |

3. Plot the results on the graph below:



4. Calculate the Difference Gain:

$$\text{DIFFERENCE GAIN} = \frac{\triangle V_{out}}{\triangle (V_A - V_B)} =$$

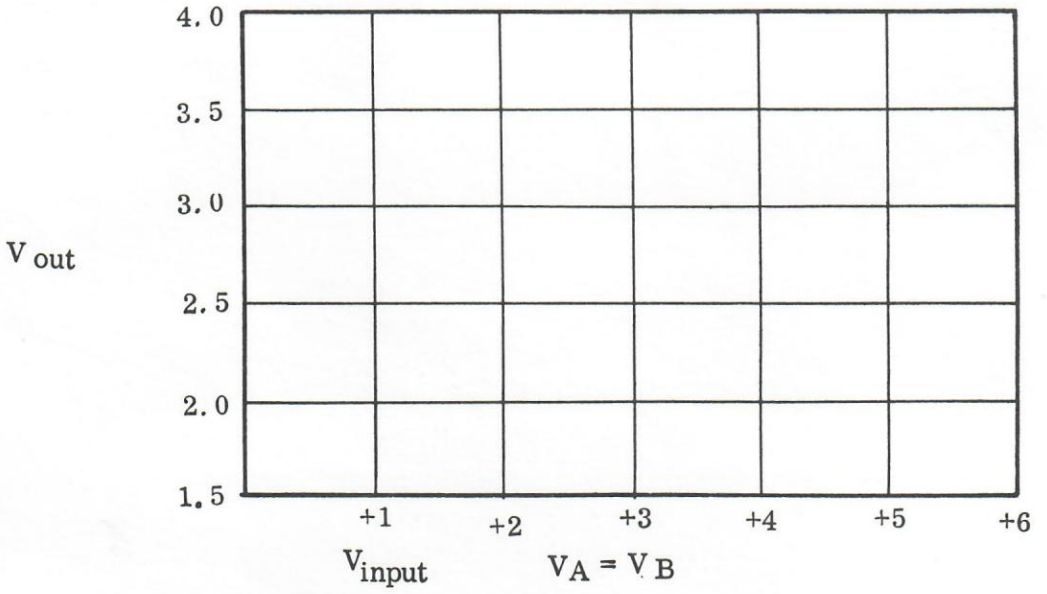
| DIFFERENCE GAIN = |
|---|

5. The COMMON MODE refers to the voltages at inputs A and B changing in COMMON. IF $V_A$ = 2 volts and $V_B$ = 2 volts, there is zero difference. If $V_A$ = 1 volt and $V_B$ = 1 volt there is zero difference and the Amplifier Output should be the same. But as both inputs increase in Common, the output voltage decreases because the emitter current increases which also increases the collect current. The COMMON MODE GAIN refers to the change in output voltage when both inputs change together. It is desirable to have a ZERO Common Mode Gain.

To measure the Common Mode Gain, set the 10K POT to the Mid-value. Connect the "center" or wiper of the 1K POT directly to the center of the 10K POT. You can now vary the 1K POT which will vary both inputs A and B. Vary the inputs (1K POT) and read the input and output:

| $V_A = V_B$ | |
|---|---|
| 0 | |
| 0.5 | |
| 1.0 | |
| 1.5 | |
| 2.0 | |
| 2.5 | |
| 3.0 | |
| 4.0 | |

6. Plot the results below:




7. Calculate the Common Mode Gain:

$$\text{COMMON MODE GAIN} = \frac{\triangle V_{out}}{\triangle V_A} =$$

| Common Mode GAIN = |
|---|

8. The RATIO of the Difference Gain to the Common Mode Gain is an indication of how good the amplifier is (higher is better) and is called the COMMON MODE REJECTION RATIO (CMRR).

Calculate the CMRR.

$$\text{CMRR} = \frac{\text{Difference Gain}}{\text{Common Mode Gain}} =$$

| CMRR = |
|---|

This number is very low. Improved CMRR circuits are discussed in the next experiment.

| Ed-Lab | CONSTANT CURRENT DIFFERENCE AMPLIFIER | EXP. NO. 21<br>Page No. 1 |
|---|---|---|

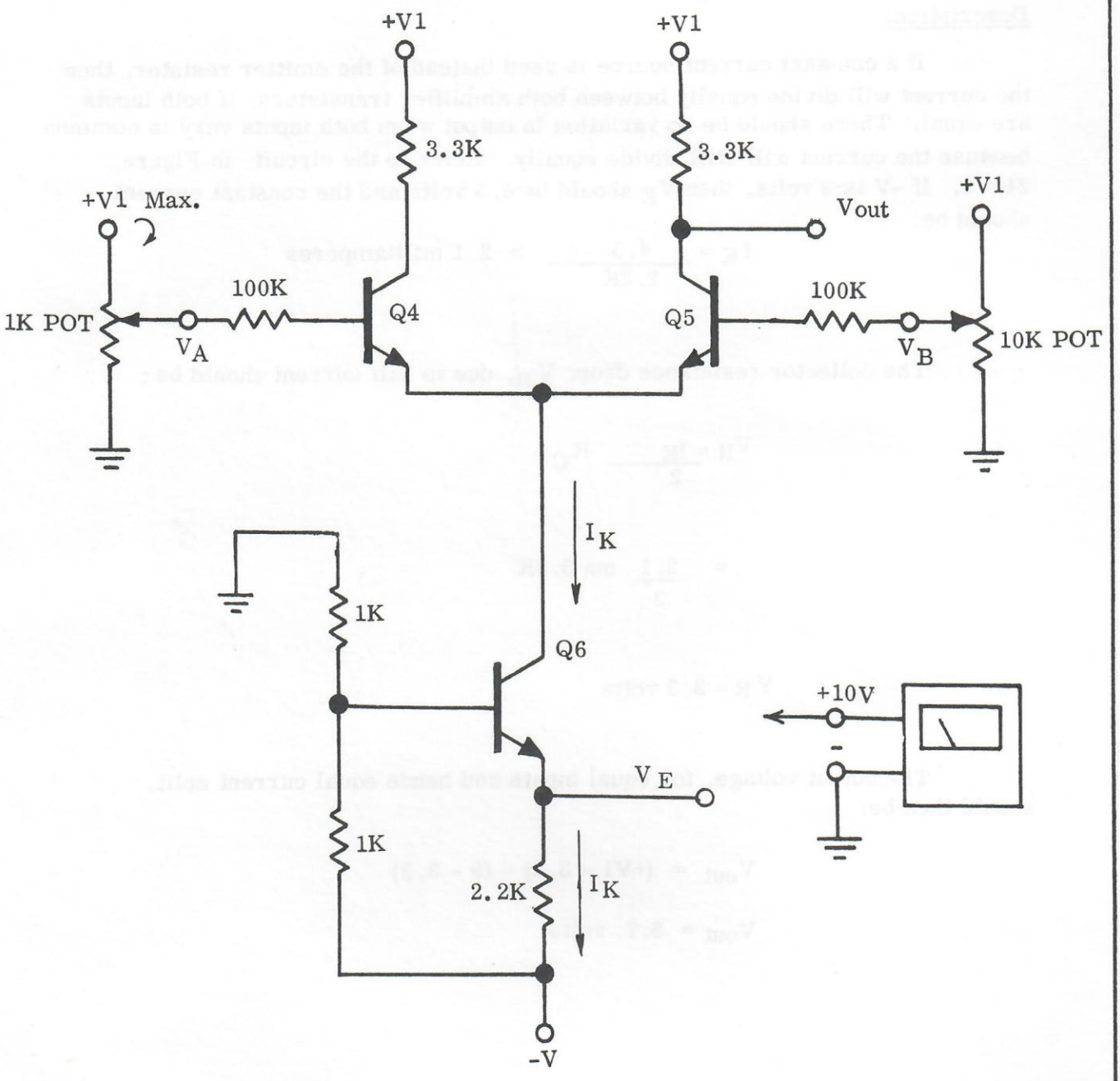



Figure 21-1 Constant Current Difference Amplifier

Objective:

You will use a constant current source instead of the emitter resistor in a Difference Amplifier. You will then measure the Gain and the CMRR.

Description:

If a constant current source is used instead of the emitter resistor, then the current will divide equally between both amplifier transistors, if both inputs are equal. There should be no variation in output when both inputs vary in common because the current will still divide equally. Refer to the circuit in Figure 21 - 1. If -V is-9 volts, then $V_E$ should be-4.5 volts and the constant current should be:

$$I_K = \frac{4.5}{2.2K} = 2.1 \text{ milliamperes}$$

The collector resistance drop, $V_R$, due to half current should be:

$$V_R = \frac{I_K}{2} R_C$$

$$= \frac{2.1}{2} \text{ ma } 3.3K$$

$$V_R = 3.3 \text{ volts}$$

The output voltage, for equal inputs and hence equal current split, should then be:

$$V_{out} = (+V1 - 3.3) = (9 - 3.3)$$

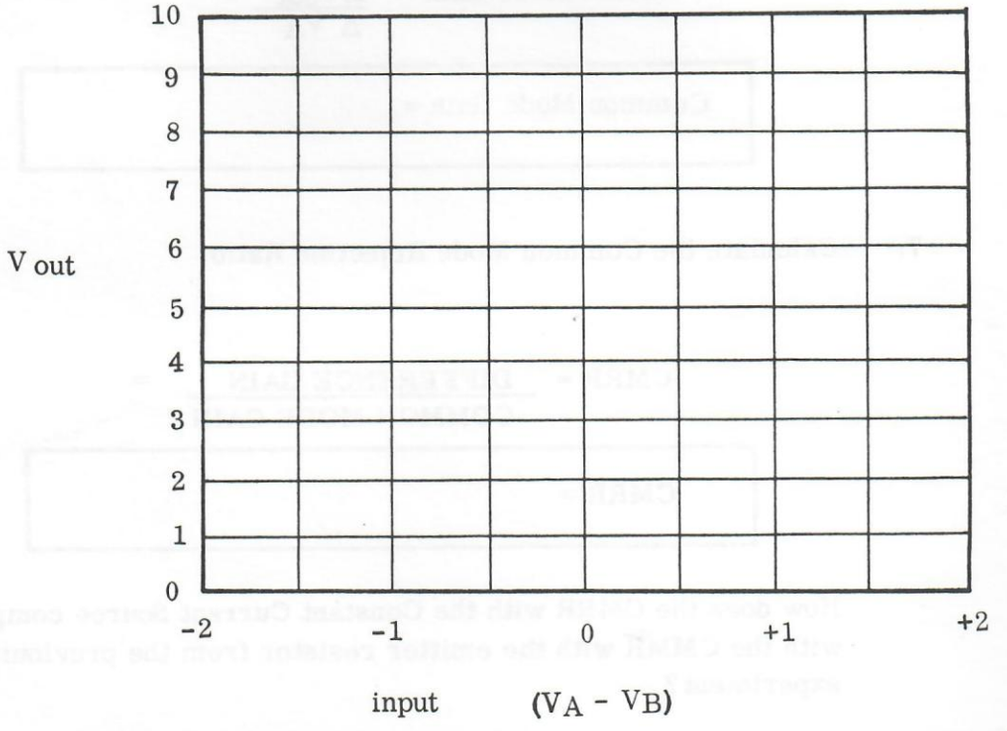
$$V_{out} = 5.7 \text{ volts}$$

Procedure:

1. Wire the circuit shown in Figure 21 - 1.
2. Vary $V_A$ and $V_B$ and measure the Difference Gain. Enter the results in the table on the following page:

| $V_A$ | $V_B$ | Vout | $(V_A - V_B)$ |
|---|---|---|---|
| 0 | 0 | | 0 |
| 0 | 0.5 | | -.5 |
| 0 | 1 | | -1 |
| 0 | 1.5 | | -1.5 |
| .5 | 0 | | +.5 |
| 1 | 0 | | +1 |
| 1.5 | 0 | | +1.5 |
| .5 | .5 | | 0 |
| .5 | 1.0 | | -.5 |
| 1 | .5 | | +.5 |
| 1 | 1 | | 0 |

3. Plot the results below:



4. Calculate the Difference Gain:

$$\text{DIFFERENCE GAIN} = \frac{\Delta V_{out}}{\Delta V_A - V_B} =$$

| DIFFERENCE GAIN = |
|---|

5. Remove the $V_B$ lead from the 10K POT and connect it to the center of the 1K POT so that $V_A = V_B$. Vary $V_A$ and measure the input and output to determine the Common Mode Gain.

| $V_A = V_B$ | $V_{out}$ |
|---|---|
| 0 | |
| 1 | |
| 2 | |
| 3 | |

In order to measure Output Voltage variations, you may use the potentiometric method (Unit 1, Exp. 14)

6. Calculate the Common Mode Gain.

$$\text{Common Mode Gain} = \frac{\Delta V_{out}}{\Delta V_A} =$$

| Common Mode Gain = |
|---|

7. Calculate the Common Mode Rejection Ratio:

$$\text{CMRR} = \frac{\text{DIFFERENCE GAIN}}{\text{COMMON MODE GAIN}} =$$

| CMRR = |
|---|

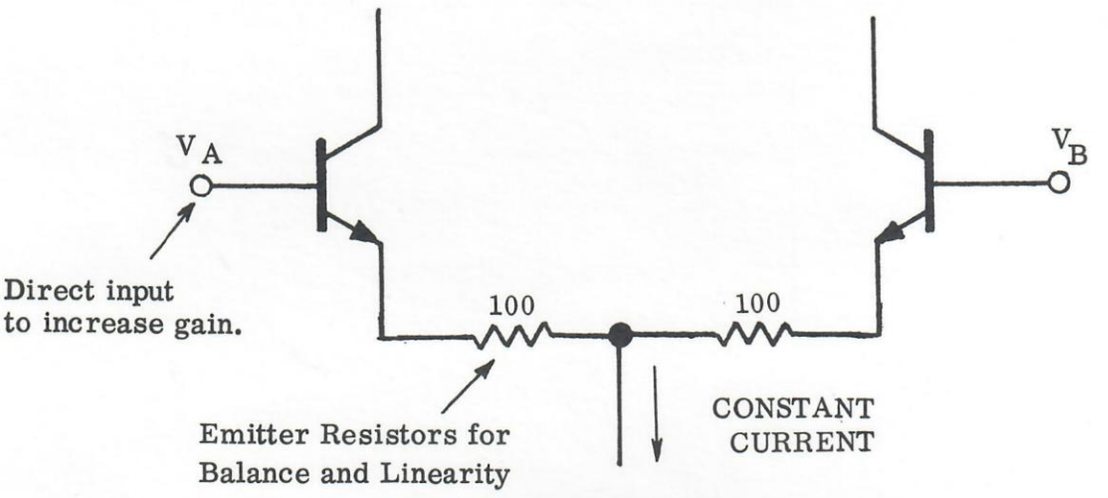
How does the CMRR with the Constant Current Source compare with the CMMR with the emitter resistor from the previous experiment?

High Gain

The Gain is increased by decreasing the Base Resistor, which is 100K in this experiment. The gain can also be increased by adding additional stages of amplification. The input impedance can be increased by having input emitter followers .

The High Gain Difference Amplifier has many applications when used with "feedback" and is called an Operational Amplifier and is used in Computers, Signal Processing, Radios, Television and Instrumentation. Applications are given in another Experiment Unit.

In many cases, two emitter resistors are inserted in the difference amplifier as shown below:




These resistors decrease the Gain due to Emitter Degeneration, but they have the effect of balancing the transistors and increasing the "linearity" of the amplifier. Base resistors are generally not used with a resulting high gain. Any desired reduced gain is achieved by using feedback.

| Ed-Lab | DARLINGTON AMPLIFIER | EXP. NO. 22<br>Page No. 1 |
|---|---|---|

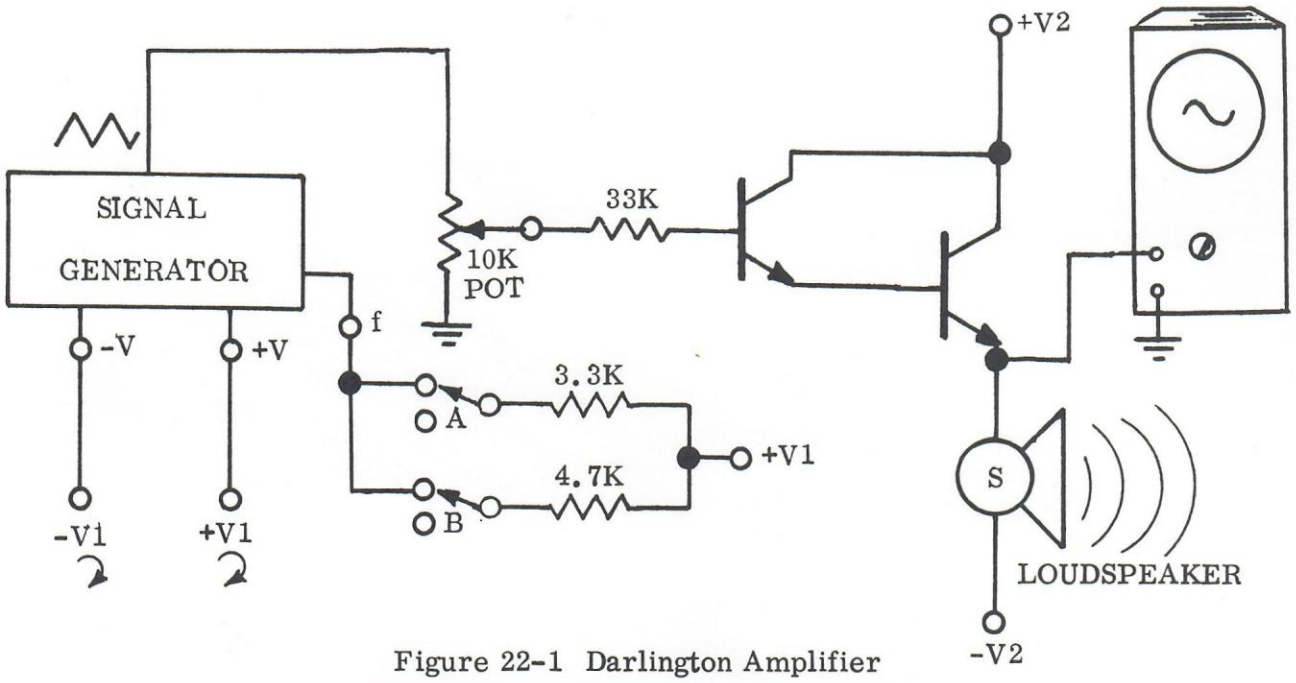



Figure 22-1 Darlington Amplifier

Objective:

You will use two transistor Emitter Followers in Cascade in a circuit called a Darlington Amplifier.

Description:

The Emitter Follower is used to provide a high input impedance and a low output impedance to "drive" output motors, loudspeakers, or inductors such as television deflection coils. In many cases, the load impedance, $R_L$, is very small, such as in a 4 OHM loudspeaker, and the emitter follower, with a BETA of 100, would provide an input impedance of ($\beta \times R_L$)400 ohms which may still be too small. Two Emitter Followers in Cascade result in an input impedance of ($\beta \times \beta \times R_L$) 40,000 OHMS. The series Base Resistor in the circuit demonstrates that there is a high circuit input impedance.

Procedure:

1. Wire the Darlington Circuit in Figure 22-1. The Signal Generator provides a Triangle Waveform which can be observed on the oscilloscope at the output loudspeaker.
2. Adjust the input signal level (10K POT) for an undistorted output as viewed on the oscilloscope.
3. Operate switches A and B which vary the voltage at "f" and thus change the frequency to the "audio-range", and listen to the sounds.

| Ed-Lab | AUDIO DRIVER | EXP. NO. 23<br>Page No. 1 |
|---|---|---|

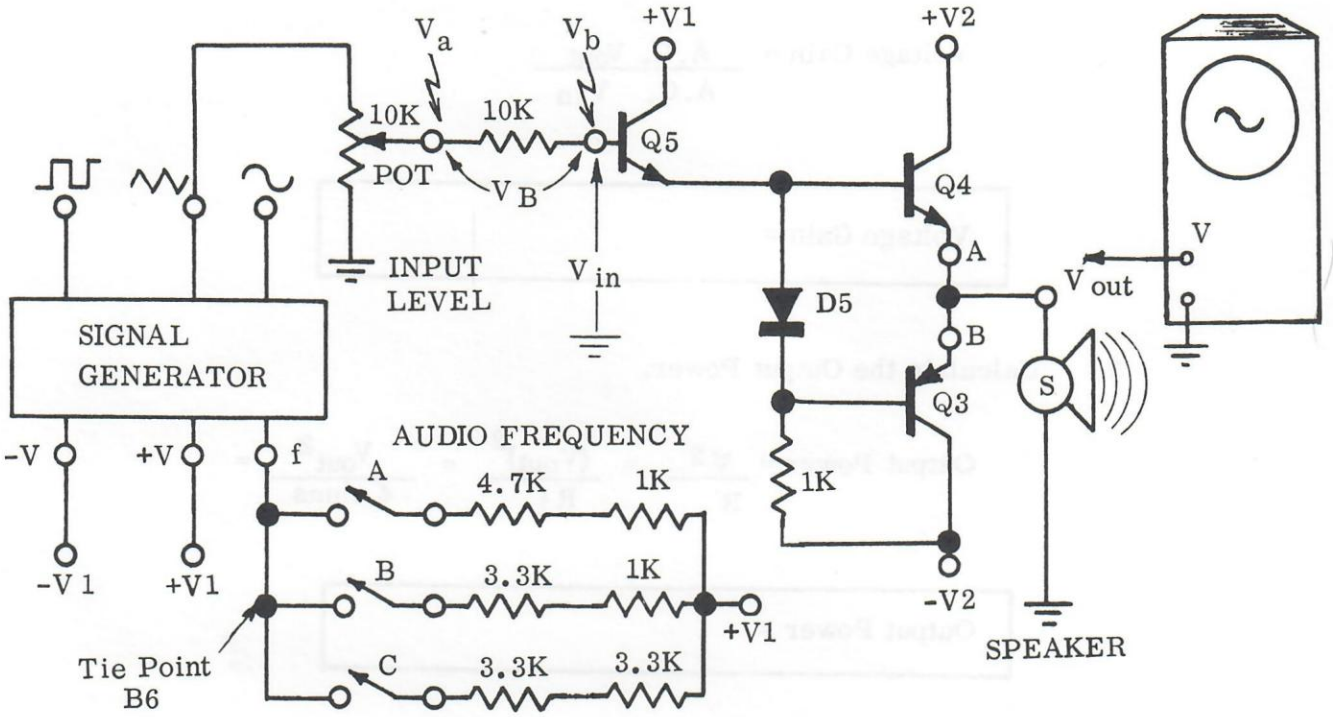



Figure 23-1 Complementary Driver

Objective:

You will use an NPN and PNP Emitter Follower as a COMPLEMENTARY OUTPUT EMITTER FOLLOWER to drive a loudspeaker.

Procedure:

1. Wire the circuit in Figure 23-1. Set all Switches UP.

   The diode is used to provide a small voltage difference between the two base voltages to compensate for the small BASE-EMITTER Voltage drops and prevent "cross-over distortion" by permitting both transistors to conduct.

2. Set the input level (10K) to result in a non-distorted triangle output.

3. Operate the Switches to generate different audio frequencies.

4. The Emitter Follower has a gain of "less than one".

   The Driver and Output stages are actually Darlington Amplifiers with one common first stage. Measure the Gain of this power output stage by using the Oscilloscope:

| A.C. $V_{in}$ = |
|---|
| A.C. $V_{out}$ = |

$$\text{Voltage Gain} = \frac{\text{A.C. } V_{out}}{\text{A.C. } V_{in}}$$

| Voltage Gain = |
|---|

5. Calculate the Output Power.

$$\text{Output Power} = \frac{V^2}{R} = \frac{(V_{out})^2}{R_L} = \frac{V_{out}^2}{4 \text{ ohms}} =$$

| Output Power = |
|---|

6. Calculate the Input Power. Measure the voltage across the series Base Resistor using the Oscilloscope:

| A.C. $V_B = (V_a - V_b) =$ |
|---|

The input current is $I = \frac{E}{R}$

$$\text{A.C. } I = \frac{(\text{A.C. } V_B)}{10K}$$

The Input Power is:

$$\text{Input Power} = VI = (\text{A.C. } V_{input})(\text{A.C. } I)$$

$$= \frac{(\text{A.C. } V_{input})(\text{A.C. } V_B)}{10,000}$$

$$= \frac{(V \text{ input})(\text{A.C. } V_B)}{10} \text{ milliwatts}$$

| Input Power = |
|---|

7. Calculate the Power Gain:

$$\text{Power Gain} = \frac{\text{Output Power}}{\text{Input Power}} =$$

| Power Gain = |
|---|

| Ed-Lab | FEEDBACK AMPLIFIER | EXP. NO. 24<br>Page No. 1 |
|---|---|---|

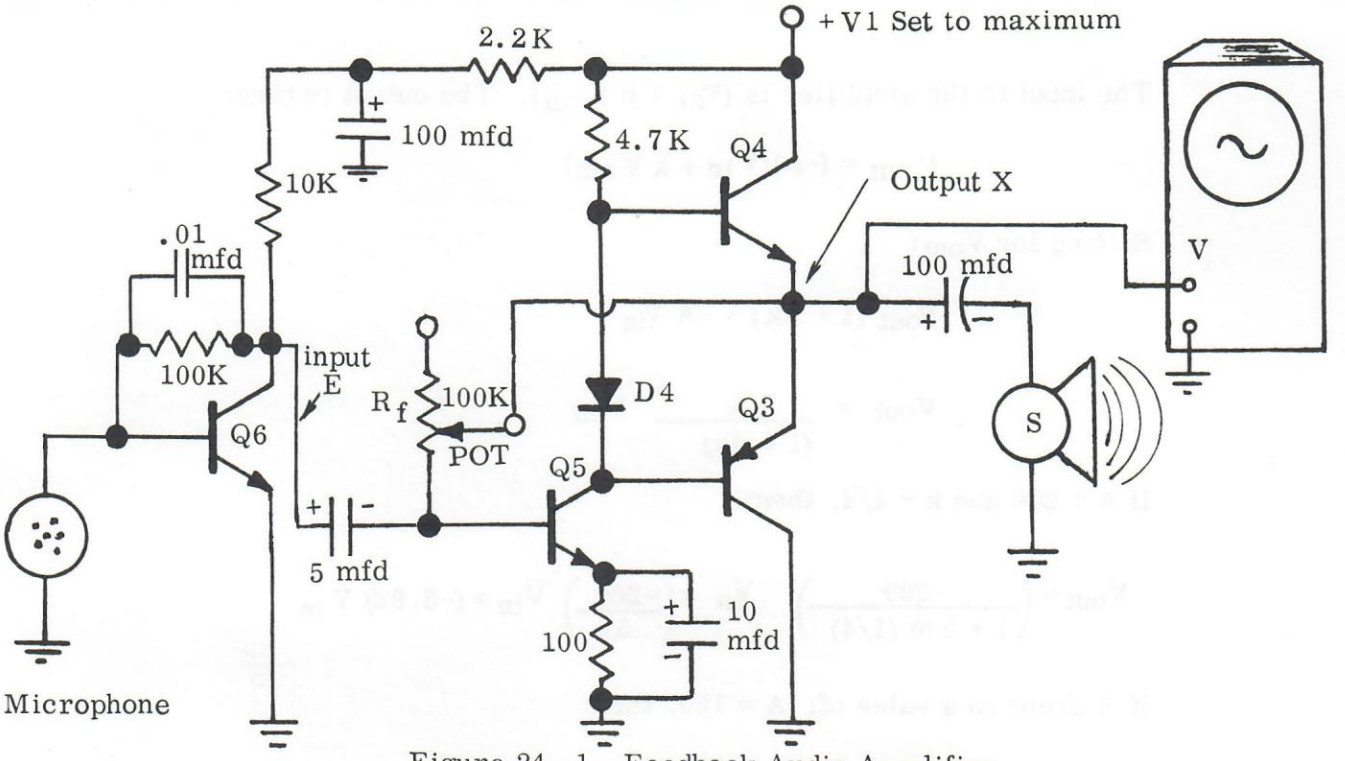



Figure 24 - 1 Feedback Audio Amplifier

Objective:

You will operate an Audio Amplifier with a Complementary Emitter Follower Output Stage and a Driver Stage with Gain. Feedback is used to stabilize the Gain and Bias and a Microphone Input is used.

Description:

The amplifier shown above has a Complementary Emitter Follower output to drive the Loudspeaker. The Driver Stage is a Common Emitter Amplifier with Gain. Whenever an amplifier with gain is used, it is necessary to Bias the circuit to operate in the linear range and to prevent distortion and clipping. The Amplifier Gain is dependent on Beta; the Beta of different transistors vary and thus the amplifier gain varies. A large gain can cause distortion and clipping. Feedback is used with high gain amplifiers to provide a constant bias and a constant gain independent of Beta. Consider an amplifier with a gain of (-A) where A can be 200 or so. Consider an amplifier with an input, $V_{in}$, and an additional feedback consisting of a voltage divider input (constant k) from the output voltage.

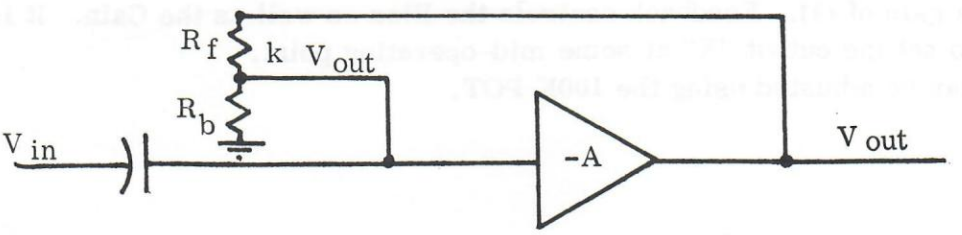



Figure 24 - 2 Amplifier with Feedback

The input to the amplifier is ($V_{in} + k V_{out}$). The output is then:

$$V_{out} = (-A)(V_{in} + k V_{out})$$

Solving for $V_{out}$:

$$V_{out}(1 + Ak) = -A V_{in}$$

$$V_{out} = \frac{-A}{(1 + Ak)} V_{in}$$

If A = 200 and k = 1/4, then:

$$V_{out} = \left(\frac{-200}{1 + 200\ (1/4)}\right) V_{in} = \left(\frac{-200}{51}\right) V_{in} = (-3.92)\ V_{in}$$

If A drops to a value of: A = 160, then:

$$V_{out} = \left(\frac{-160}{1 + 160\ (1/4)}\right) V_{in} = \left(-\frac{160}{41}\right) V_{in} = (-3.90) V_{in}$$

Note that the amplifier gain, A, has dropped from 200 to 160 or by 20 percent while the overall circuit gain has dropped from -3.92 to -3.90 or by only (.02) in 3.9 or 1/2 percent.

If A and Ak is much larger than 1, then 1+Ak is approximately equal to Ak. Then:

$$V_{out} \cong \frac{-A}{+Ak} V_{in}$$

$$V_{out} \cong -\frac{1}{k} V_{in}$$

The circuit gain is the "reciprocal" of the feedback. A feedback of (1/4) results in a gain of (4). Feedback controls the Bias as well as the Gain. It is desirable to set the output "X" at some mid-operating point. Feedback can be adjusted using the 100K POT.

Procedure:

1. Wire the circuit shown in Figure 24 - 1. The Microphone is the 1000 ohm earphone.
2. Set the 100K feedback POT at 1/2 full scale.
3. Speak into the microphone. Monitor the output at point "X" with the oscilloscope.
4. Record the peak-to-peak output voltage

| $V_{output}$ (peak-to-peak) = |
|---|

5. Place the microphone against the speaker. The amplifier should begin to oscillate due to the acoustic feedback from the speaker to the microphone.
6. Record the peak-to-peak voltage of the oscillation.

| $V_{oscillation}$(peak to peak) = |
|---|

7. Adjust the 100K feedback POT until the oscillation STOPS.
8. Speak into the microphone and record the peak-to-peak output Voltage.

| $V_{output}$ (point X) (peak-to-peak) = |
|---|

NO OSCILLATION

9. Note that the Output Voltage will be smaller after the feedback has been adjusted to prevent oscillation. This is because the gain of the amplifier has been reduced by increasing the amount of negative feedback.
10. Record the resistance of the feedback component (used portion of the 100K POT) when the oscillation is eliminated

| $R_f$ = |
|---|

11. Speak into the microphone.
Record the input voltage at point E.

| $V_{in}$ (point E) peak-to-peak = |
|---|

12. Calculate the amplifier gain

FOR NO OSCILLATION

$$\text{Gain} = \frac{V_{\text{output (PointX) (peak-to-peak)(NO OSCILLATION)}}}{V_{in} \text{ (Point E)(peak to peak)}}$$

| GAIN = |
|---|

13. Calculate the amplifier maximum gain

$$\text{MAX GAIN} = \frac{V_{\text{out (STEP 4)}}}{V_{in} \text{ (STEP 11)}} =$$

| MAX GAIN = |
|---|

| Ed-Lab | FET OPERATION | EXP. NO. 25<br>Page No. 1 |
|---|---|---|

Use of an external Digital Multimeter is preferable. If none is available use Ed-Lab meter.

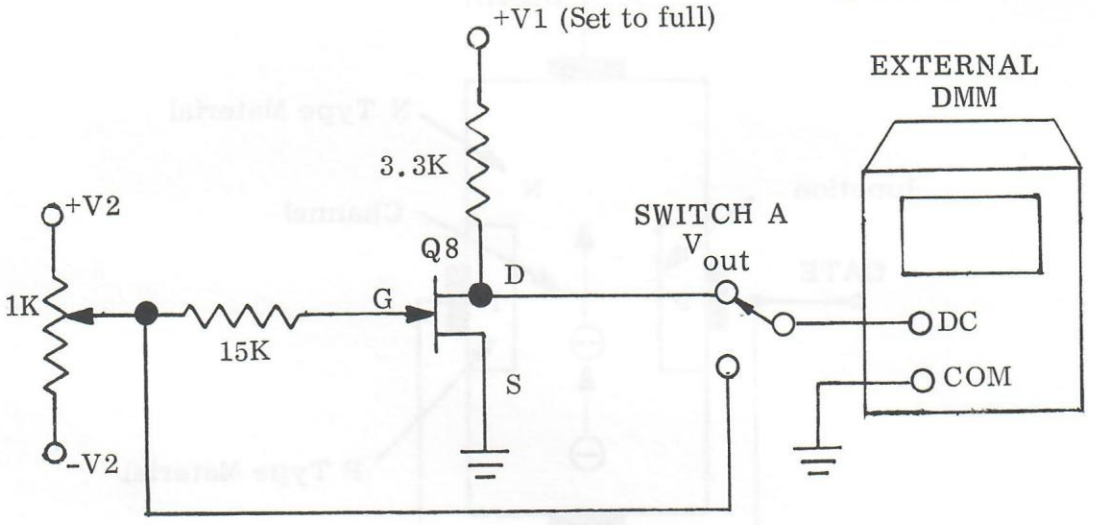



Figure 25-1 FET D.C. Amplifier

Objective:

You will test an FET Transistor with a D.C. Input and measure the Input and Output voltages. The D.C. Gain and the correct operating point for the FET will be determined. The value of the Transconductance of the FET will be calculated.

Description:

The Junction Field Effect Transistor (JFET) consists of a bar of N type semiconductor material with a section of P type material surrounding the center of the bar. The three JFET terminals, Gate, Drain and Source are connected as indicated in the diagram in Figure 25-1. A sketch of the JFET is shown in Figure 25-2 on the following page. In normal operation the Gate is Negative. This negative voltage biases the semiconductor junction between the P type Gate and the N type Channel in the reverse direction. When the junction is thus reverse biased, the current flow is controlled. The negative gate voltage repels the free electrons in the N type material toward the center and out of the N type material near the Gate. This creates a Depletion Layer in the N type material in the region of the gate and causes the resistance of the material to increase. This change in resistance to current flow from Drain to Source, which is proportional to the Gate to Source Voltage is used to generate voltage gain. This change in output current caused by changing input voltage is called the FET Transconductance (gm).

$$gm = \frac{\Delta I_{out}}{\Delta V_{in}} = \text{Transconductance}$$

The units of transconductance is $\frac{1}{\text{ohms}}$ and is called "mho" (ohm spelled backwards).

The construction of an FET is shown in Figure 25-2.

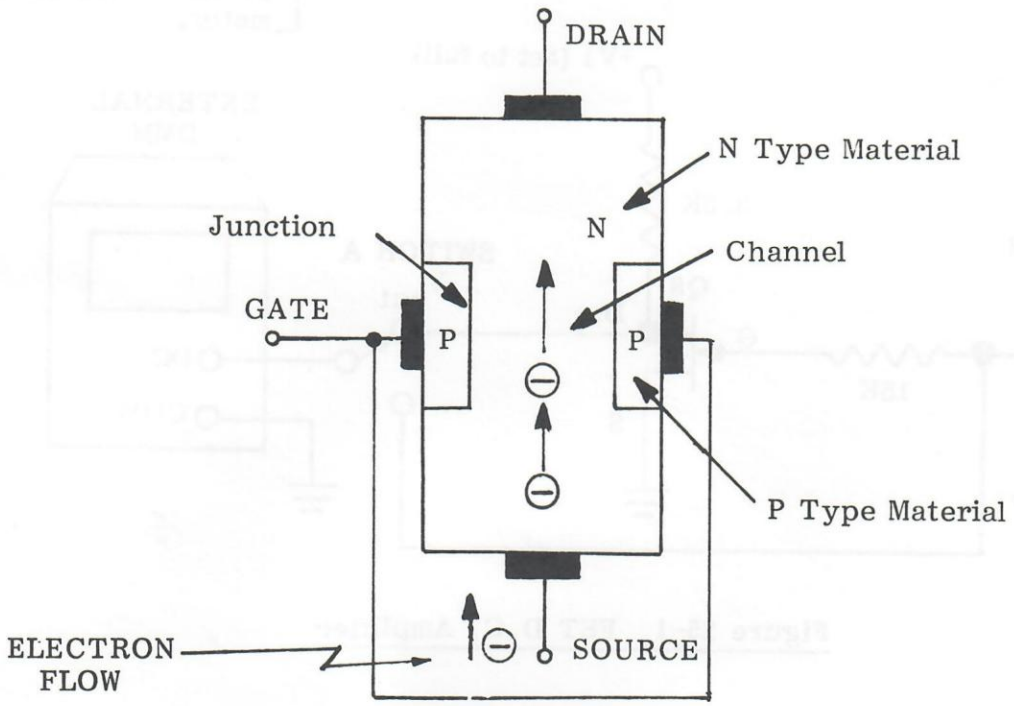



Figure 25-2 JFET Construction

The FET consists of a bar of N-type semiconductor material. At one end is a electrical contact labled DRAIN and at the other end of the bar is a contact labeled SOURCE.

The center of the bar is bracketed with two pieces of P-type material which are both connected to a contact labeled GATE. The surface where the P-type material and the N-type material touch forms the P-N junction. The region of N-type material between the gate pieces is called the CHANNEL where current can flow. Since the channel is type N material the device is called an N channel FET. The schematic symbol is shown in Figure 25-3 below:

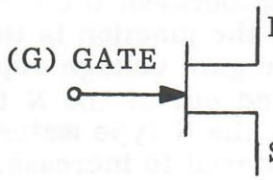



Figure 25-3 FET Schematic Symbol

When the FET Drain and Source are connected to a power supply, current will flow through the FET Channel. Since the channel is SEMICONDUCTOR MATERIAL, the channel will have a resistance and the voltage drop throughout the length of the channel will be more or less linear. The channel at the point where the gate is attached will assume a positive voltage of a fraction of the Drain Voltage. This situation is illustrated in Figure 25-4.

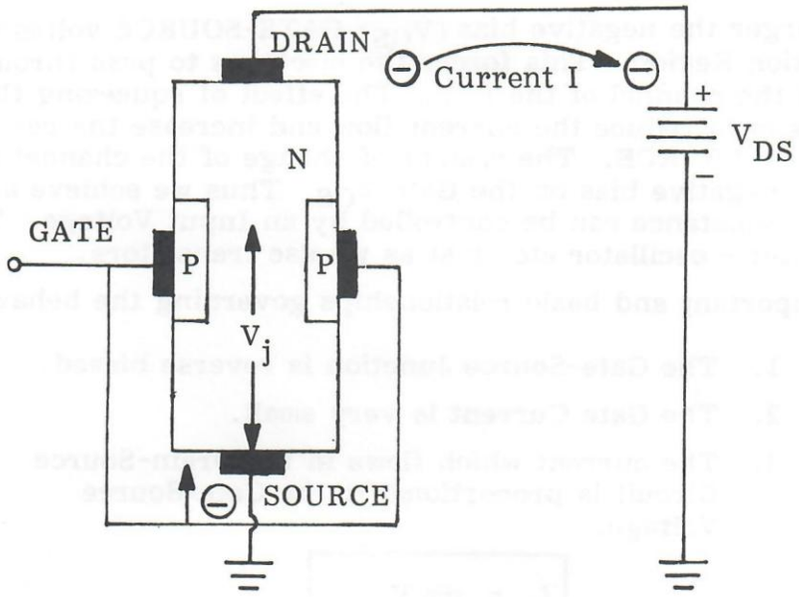



Figure 25-4 FET Channel Current and Junction Voltage

If we connect the GATE terminal to a NEGATIVE voltage source, the GATE junction can be reverse biased. The junction will then behave like a reverse biased diode. Very little current will flow and the impedance will be very high. The main function of the reverse biased gate junction will be to generate a negative potential at the Gate-Channel junction. This negative potential will repel electrons away from the junction and force them to move to the center of the channel. The region near the junction where electrons are repelled to the center of the channel is called the DEPLETION REGION. The path of the electrons in the channel is illustrated in Figure 25-5.

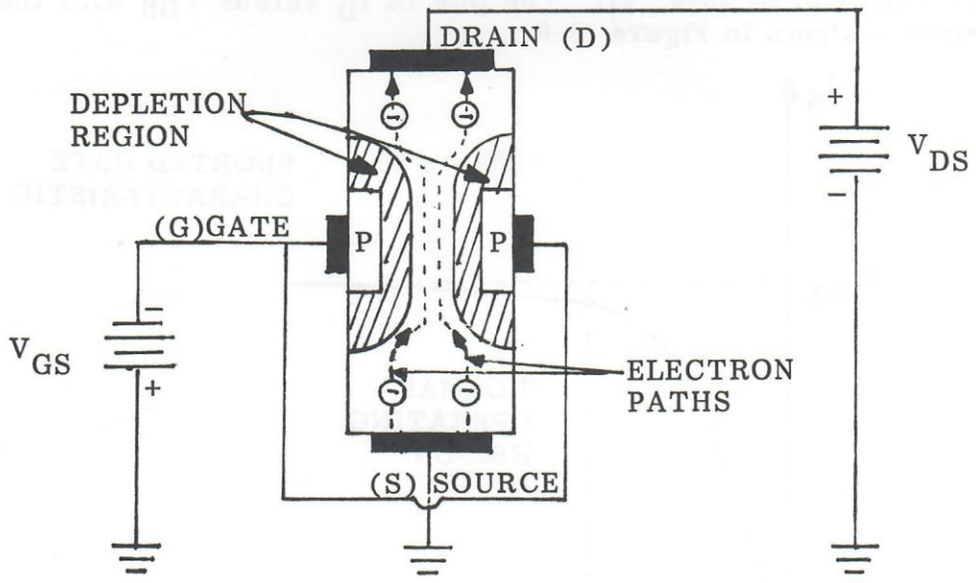



Figure 25-5 Current Flow in an FET with Depletion Region

The larger the negative bias ($V_{GS}$) GATE-SOURCE voltage applied, the larger the Depletion Region. This forces the electrons to pass through a much narrower area in the channel of the FET. The effect of squeezing the electrons together like this is to reduce the current flow and increase the resistance of the FET from DRAIN to SOURCE. The amount of change of the channel resistance is controlled by the negative bias on the Gate $V_{GS}$. Thus we achieve a device whose current flow and resistance can be controlled by an Input Voltage. This can be used as an amplifier - oscillator etc. just as we use transistors.

The important and basic relationships governing the behavior of the FET are as follow:

1. The Gate-Source Junction is reverse biased.
2. The Gate Current is very small.
3. The current which flows in the Drain-Source Circuit is proportional to the Gate-Source Voltage.

$$I_D = gm\, V_{GS}$$

$I_D$ = Source Current

gm = Transconductance

$V_{GS}$ = Gate Source Voltage

If we were to vary the Drain Source voltage $V_{DS}$ while the Gate was shorted to the Source we could measure $V_{DS}$ versus $I_D$ and plot the behavior of the Drain Source V-I curve to changes in $V_{DS}$. Notice that, when the Gate is shorted to the Source the Gate Junction will be reverse biased because of the voltage drop in the channel. This causes the channel to be positive in the region of the Gate (See Figure 25-4, $V_j$). The plot of $I_D$ versus $V_{DS}$ with the Gate shorted to the Source is shown in Figure 25-6.

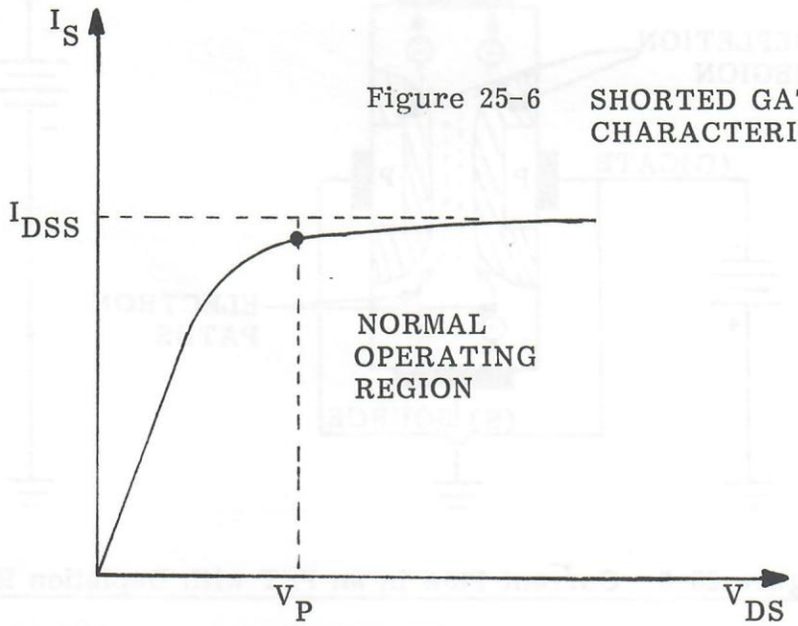



Figure 25-6 SHORTED GATE CHARACTERISTIC

Referring to Figure 25-6, as $V_{DS}$ increases from zero, the Source current increases steeply. At a voltage, $V_p$, the curve levels off and the FET provides constant current. $V_p$ is called the PINCH OFF VOLTAGE. It is the Voltage at which the depletion layers almost meet in the center of the channel. When the $V_{DS}$ is increased above the Pinch Off Voltage it causes the channel voltage in the gate region $V_j$ to increase, which further reverse biases the gate junction and restricts the current flow. Therefore, the Drain current remains nearly constant for $V_{DS}$ larger than $V_p$.

If the gate voltage is changed whiled the FET is operating above the Pinch Off Voltage the Drain current will vary with the Gate Voltage. This can be illustrated with the family of characteristic curves shown in Figure 25-7 below:

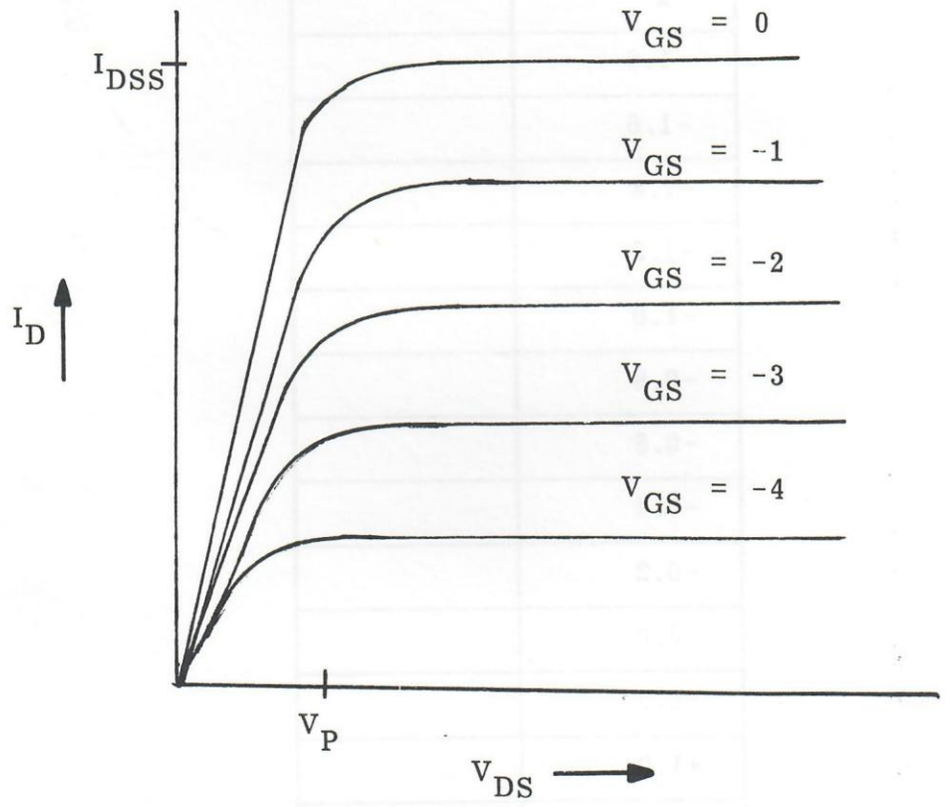



Figure 25-7 FET Characteristics

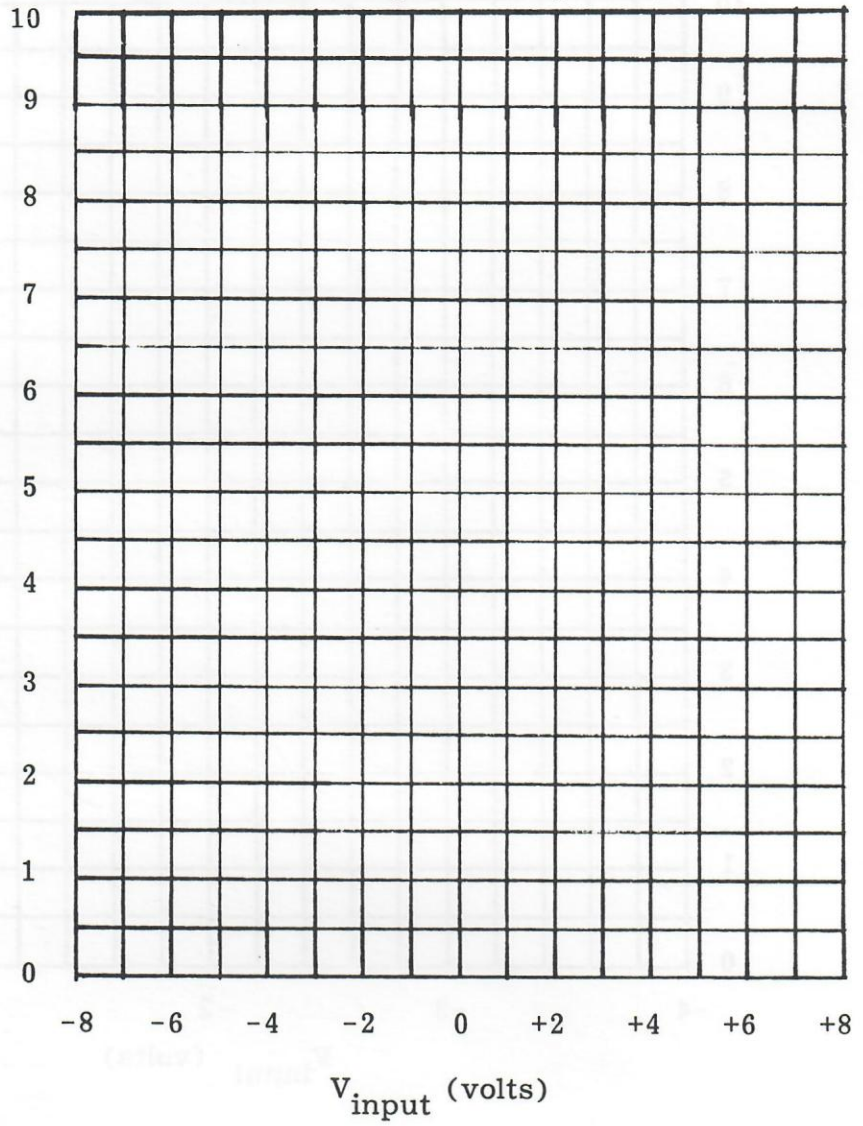
PROCEDURE:

1. Wire the circuit shown in Figure 25-1.
2. Use the D.C. Meter to measure the input and output voltages Adjust the $V_{in}$ (10K POT) to the values shown in the table and record the $V_{out}$.

| $V_{in}$ SWITCH A DOWN | $V_{out}$ SWITCH A UP |
|---|---|
| -8 | |
| -6 | |
| -4 | |
| -2 | |
| -1.8 | |
| -1.6 | |
| -1.4 | |
| -1.2 | |
| -1.0 | |
| -0.8 | |
| -0.6 | |
| -0.4 | |
| -0.2 | |
| 0.0 | |
| +0.5 | |
| +1.0 | |
| +2.0 | |
| +4.0 | |
| +6.0 | |
| +8.0 | |

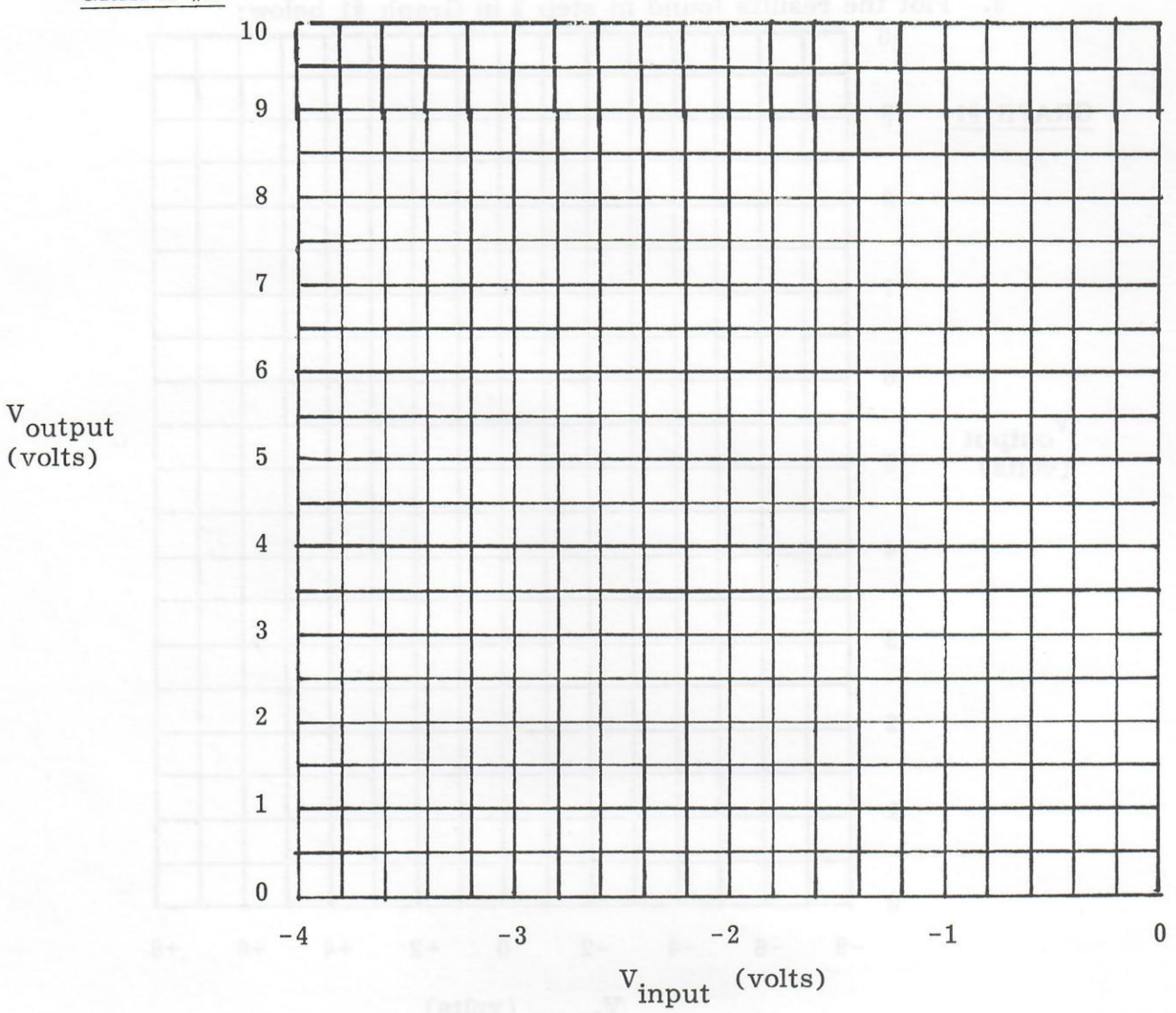
3. Plot the results found in step 2 in Graph #1 below:

GRAPH #1




Note that the Curve changes drastically only from Vin = 0 to Vin = -4 volts. Plot this part of the curve on Graph #2 on the following page.

GRAPH #2




4. Graph #2 of step 3 has a linear region where the curve has a straight slope. Label a point of the curve at the top of the linear region ($P_A$). Label a point at the bottom of the linear region ($P_B$).

   For $P_A$:

   $V_{out}$ = _____

   $V_{in}$ = _____

FOR $P_B$:

$V_{out}$ = _____

$V_{in}$ = _____

5. Calculate the mid-point of the linear region.

$$V_{out\ (mid\ point)} = \frac{V_{out\ A} - V_{out\ B}}{2} + V_{out\ B} =$$

$$\frac{(\quad) - (\quad)}{2} + (\quad) = _____$$

Label the $V_{out}$ (mid point) on the graph of step 3.
Record the Input Voltage Vin for the mid point. This mid-point would correspond to the normal input bias voltage.

$$V_{in\ (mid\ point)} = _____$$

6. Calculate the D.C. Gain of the FET Amplifier in this circuit.

$$GAIN = \frac{\Delta V_{out}}{\Delta V_{in}} = \frac{V_{out\ A} - V_{out\ B}}{V_{in\ B} - V_{in\ A}} = \frac{(\quad)}{(\quad)} =$$

| GAIN = |
|---|

7. Calculate the FET Transconductance.

$$\text{Transconductance} = gm = \frac{\Delta I_{out}}{\Delta V_{in}} = \frac{\frac{\Delta V_{out}}{R_L}}{\Delta V_{in}} = \frac{GAIN}{R_L}$$

Note that transconductance has the units of $\frac{1}{ohm}$ or mho

$$gm = \frac{GAIN\ (of\ Step\ 6)}{R_L} = \frac{(\quad)}{3.3K} = _____$$

| gm = |
|---|

gm is usually written in terms of MICROMHOS. The answer should be on the order of 1000 micromhos.

QUESTIONS:

1. Since the Gate Junction is reverse biased in the linear region of operation of the FET, would you say the gate impedance is high or low?

2. Can the current in the FET Drain - Source Circuit be turned nearly OFF with the gate?

3. Can the FET be used as a Solid-State Switch?

4. Which portions of the graph show that the FET can be used as a Switch?

5. What would the gain of this amplifier be if the load resistor was changed to 10K?

6. At what $V_{GS}$ voltage would you normally bias the FET Amplifier in this experiment?

| Ed-Lab | FET COMMON SOURCE AMPLIFIER | EXP. NO. 26<br>Page No. 1 |
|---|---|---|

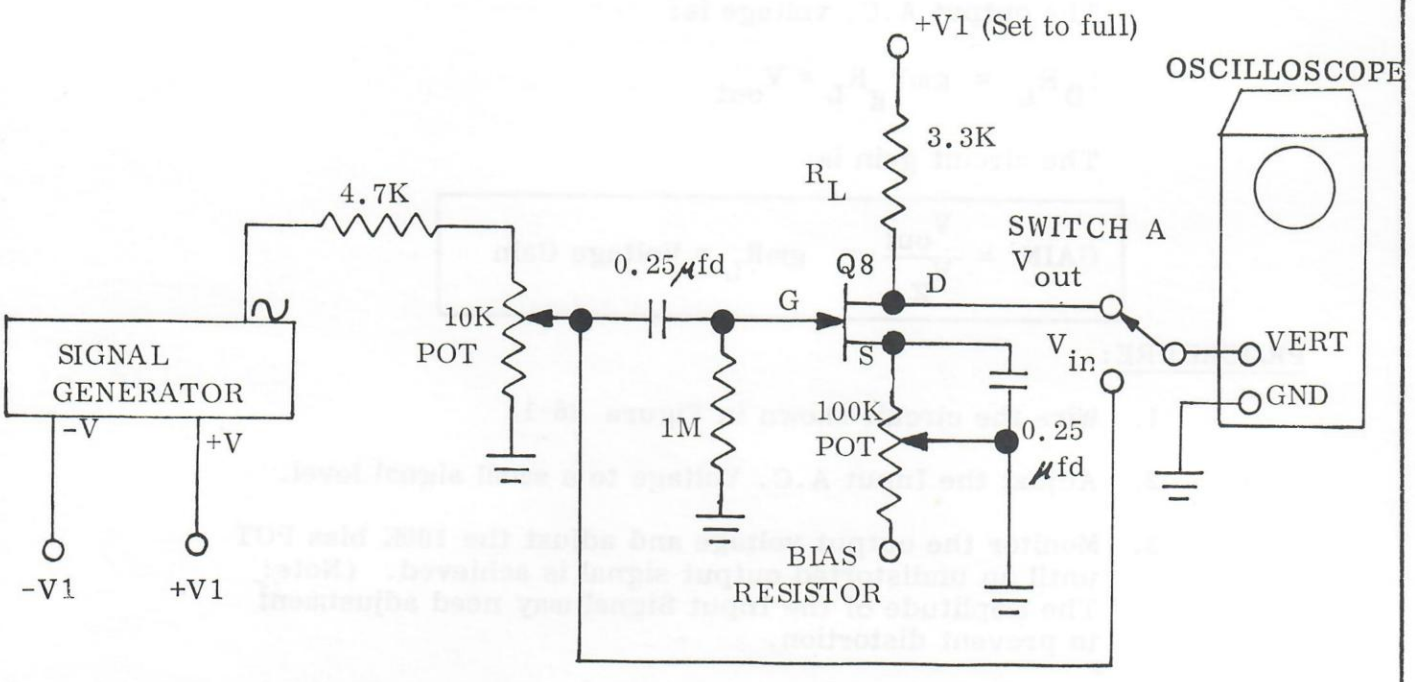



Figure 26-1 Common Source FET A.C. Amplifier

Objective:

You will use an FET in a Common Source Amplifier and measure the A.C. Amplifier Voltage Gain.

Description:

The FET is used as a Common Source Amplifier. This amplifier is an A.C. Voltage Amplifier. The A.C. input signal is applied to the FET Gate. Voltage variation at the gate causes a change in the Drain to Source Current in the FET. The current variation generates an output voltage across $R_L$ and the input signal is amplified. Note that the Source path to ground for D.C. is through the 100K POT. The bias current in the FET develops a positive voltage at the Source. This serves to bias the Gate junction in reverse, placing the FET in the region of linear operation measured in Experiment No. 25. The 0.25μfd capacitor across the bias POT provides a low impedance path to ground for A.C. signals to prevent Source Degeneration. This is exactly the same principle as the Emitter Bypass Capacitor usage in the Common Emitter Transistor circuit.

The current in the FET Drain - Source circuit is given by:

$I_D = gmV_g$

$I_D$ = DRAIN CURRENT
$V_g$ = GATE VOLTAGE
gm = TRANSCONDUCTANCE

The output A.C. voltage is:

$$I_D R_L = gmV_g R_L = V_{out}$$

The circuit gain is:

$$GAIN = \frac{V_{out}}{V_g} = gmR_L = \text{Voltage Gain}$$

PROCEDURE:

1. Wire the circuit shown in Figure 26-1.
2. Adjust the Input A.C. Voltage to a small signal level.
3. Monitor the output voltage and adjust the 100K bias POT until an undistorted output signal is achieved. (Note: The amplitude of the Input Signal may need adjustment to prevent distortion.
4. Record the Input Voltage and Output Voltage Amplitude (peak to peak)

| $V_{input}$ = |
|---|
| $V_{output}$ = |

5. Calculate the voltage Gain from the data in step 4 above.

$$GAIN\ (voltage) = \frac{V_{out}}{V_{in}} = \frac{(\quad)}{(\quad)} =$$

6. Calculate the Theoretical Gain from the equation given in the Description. Use the gm found in Experiment No. 25.

gm (from Experiment No. 25) =

GAIN (voltage) = $gmR_L$ = ( ) ( ) =

$GAIN_V$ =

7. How does the theoretical gain calculated in step 6 compare to the actual gain measured in step 5?
8. Increase the Input Signal Amplitude until the Output Signal just begins to distort. Record the Maximum Output Amplitude.

Output Amplitude max. =

QUESTIONS:

1. How does the voltage gain of the FET A.C. Common Source Amplifier from this experiment compare to the gain of a similar Common Emitter Transistor D.C. Amplifier experiment?

2. Is the Gate Junction forward biased or reverse biased in the normal operating range?

3. To what Transistor Amplifier circuit is the FET Common Source Amplifier similar?

| Ed-Lab | FET COMMON DRAIN AMPLIFIER | EXP. NO. 27<br>Page No. 1 |
|---|---|---|

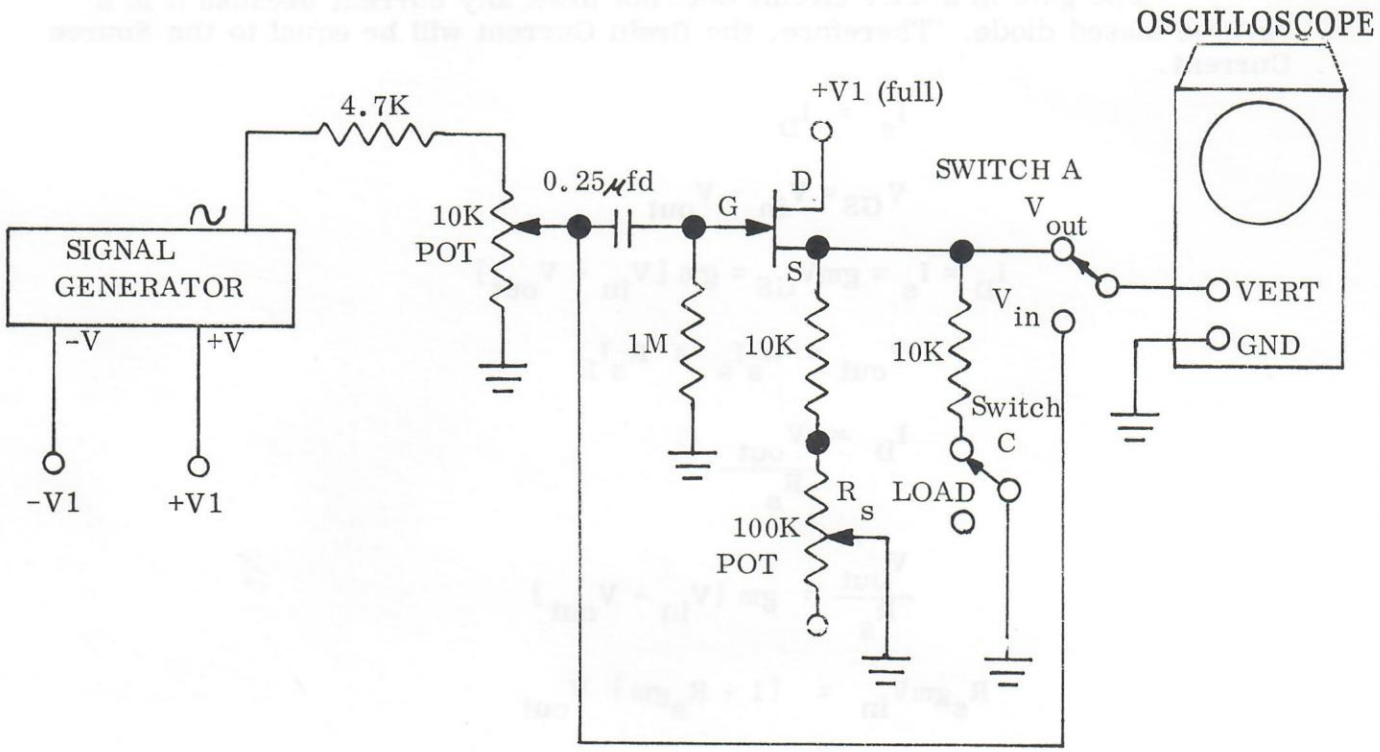



Figure 27-1 Common Drain Amplifier

Objective:

You will build and test a Common Drain FET Amplifier and measure the circuit Gain and Output Impedance.

Description:

This circuit is the FET equivalent of the Common Collector or Emitter Follower Transistor Amplifier. We can calculate the voltage gain and current gain using loop voltage equations.

Examine the common Drain circuit shown in Figure 27-2.

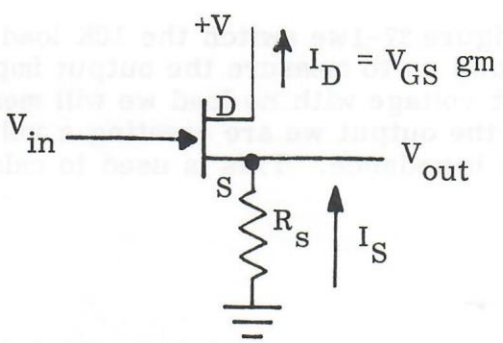



Figure 27-2 Common Drain Amplifier

The gate in a FET circuit does not draw any current because it is a reverse biased diode. Therefore, the Drain Current will be equal to the Source Current.

$$I_s = I_D$$

$$V_{GS} = V_{in} - V_{out}$$

$$I_D = I_s = gmV_{GS} = gm\,[V_{in} - V_{out}]$$

$$V_{out} = R_s I_s = R_s I_D$$

$$I_D = \frac{V_{out}}{R_s}$$

$$\frac{V_{out}}{R_s} = gm\,[V_{in} - V_{out}]$$

$$R_s gm V_{in} = [1 + R_s gm]\, V_{out}$$

$$GAIN = \frac{V_{out}}{V_{in}} = \frac{R_s gm}{1 + R_s gm} = \frac{1}{1 + \frac{1}{R_s gm}}$$

$$\boxed{GAIN = \frac{1}{1 + \frac{1}{R_s gm}}}$$

Typically for $R_s$ = 10K, gm = 1000micromhos

$R_s gm = 10x10^3 \times 1000x10^{-6} = 10^7 x10^{-6} = 10$

The Gain here is $\frac{1}{1.1}$ or .9

The gain will always be less than 1.

In the idealized FET the Gate current should be zero. In a practical FET the Gate current is very small. The gain does not vary much as the $R_S$ or load resistor varies. Therefore this circuit has a low output resistance.

In the circuit of Figure 27-1we switch the 10K load resistor ON and OFF on the output signal. This allows us to measure the output impedance of the circuit. When we measure the output voltage with no load we will measure the $V_{out}$ of circuit 27-3. When we load the output we are creating a voltage divider with the 10K load and the output impedance. This is used to calculate the value of the output impedance.

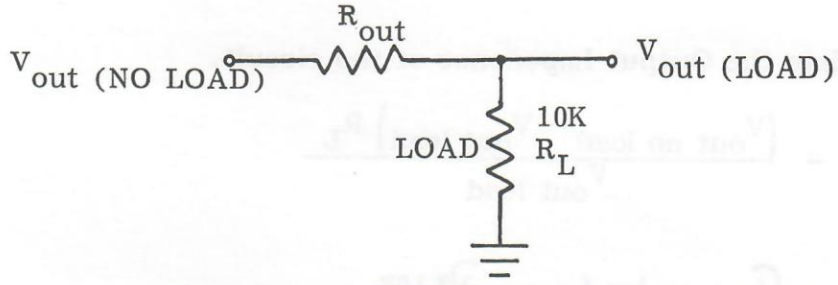



Figure 27-3 OUTPUT IMPEDANCE & LOAD

$$V_{out\ load} = \frac{V_{out\ no\ load}\ R_L}{R_L + R_{out}}$$

$$R_{out} = \frac{V_{out\ no\ load}\ R_L - V_{out\ load}\ R_L}{V_{out\ load}}$$

$$R_{out} = \frac{\left(V_{out\ no\ load} - V_{out\ load}\right) R_L}{V_{out\ load}}$$

PROCEDURE:

1. Wire the circuit shown in Figure 27-1.
2. Place Switch C DOWN (NO LOAD)
3. Adjust the Input Voltage Amplitude to a convenient level. Record the Input Voltage Amplitude and the Output Voltage Amplitude.

| $V_{input}$ = |
|---|
| $V_{output}$ = |

4. Calculate the circuit GAIN.

$$GAIN = \frac{V_{out}}{V_{in}} = \frac{(\quad)}{(\quad)} =$$

| GAIN = |
|---|

5. Place Switch C UP (10K load). Record $V_{load}$.

| $V_{load}$ = |
|---|

| Ed-Lab | FET COMMON DRAIN AMPLIFIER | EXP. NO. 27<br>Page No. 4 |
|---|---|---|

6. Calculate the Output Impedance of the circuit.

$$R_{out} = \frac{\left(V_{out\ no\ load} - V_{out\ load}\right) R_L}{V_{out\ load}}$$

$$R_{out} = \frac{\left(\quad\right) - \left(\quad\right) \times 10K}{\left(\quad\right)} = _____$$

$R_{out}$ =

QUESTIONS:

1. If we assume the FET gate draws no current (infinite impedance) what is the input impedance for the circuit in Figure 27-1.
2. Will the Current Gain of this circuit be high or low?
3. Calculate the Current Gain (assuming voltage gain of 1).

$$\text{Current Gain} = \frac{R_{in}}{R_{load}} \quad \text{where } R_{Load} = 10K$$

4. Is the gain less than 1 as expected?

| Ed-Lab | FET COMMON GATE AMPLIFIER | EXP. NO. 28<br>Page No. 1 |
|---|---|---|

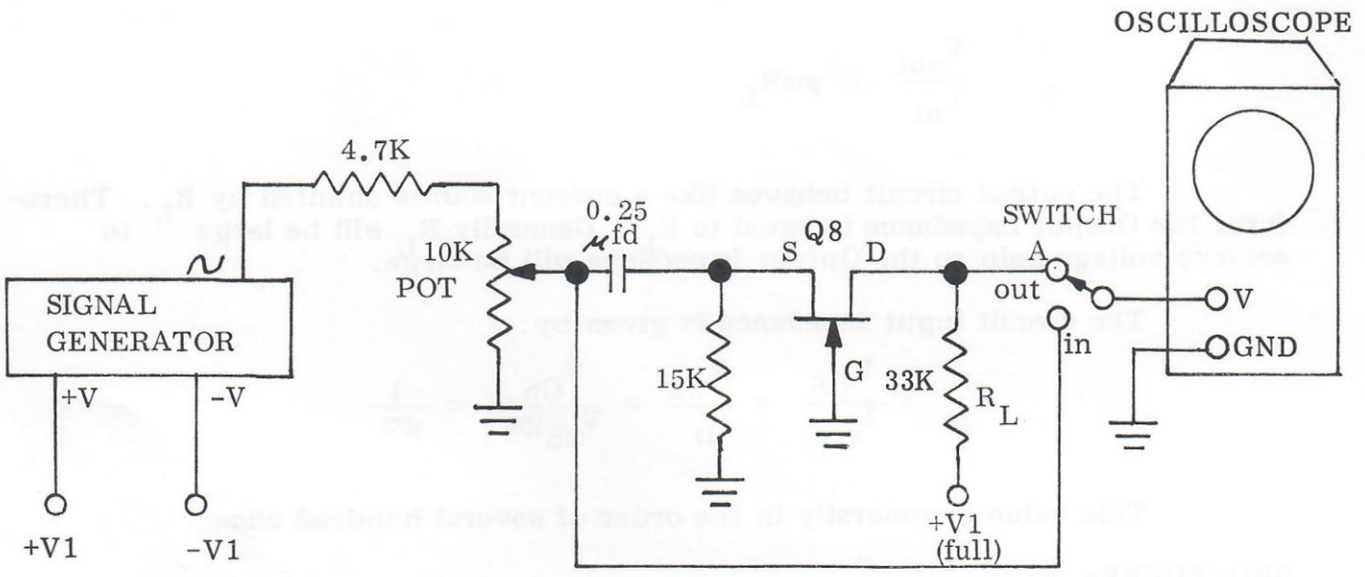



Figure 28-1 Common Gate FET Amplifier

Objective:

You will build and test a Common Gate FET Amplifier. The amplifier Gain will be measured.

Description:

In the Common Gate Amplifier, the Gate is connected to ground. The input signal is applied to the Source and the output is taken from the Drain.

This circuit is the FET equivalent of the Common Base Amplifier. The features of this circuit are low Input Impedance and high Output Impedance. The voltage and current gain are low. This circuit is used to achieve impedance matching of low impedance source to a higher impedance load and so it is particularly useful in R.F. circuits.

In Figure 28-1, note that the $V_{GS}$ Gate-Source Voltage is simply the Input Voltage. Therefore the Drain Current is given by:

$$I_D = V_{in} gm$$

The Load is connected to the Drain so the output voltage will be:

$$V_{out} = I_D R_L = V_{in} gm\ R_L$$

and the Voltage gain is:

$$\frac{V_{out}}{V_{in}} = gmR_L$$

The output circuit behaves like a current source shunted by $R_L$. Therefore, the Output Impedance is equal to $R_L$. Generally $R_L$ will be large to achieve voltage gain so the Output Impedance will be large.

The circuit input impedance is given by:

$$R_{in} = \frac{V_{GS}}{I_s} = \frac{V_{GS}}{I_D} = \frac{V_{GS}}{V_{GS}gm} = \frac{1}{gm}$$

This value is generally in the order of several hundred ohms.

PROCEDURE:

1. Wire the circuit shown in Figure 28-1.

2. Monitor the Output Voltage and adjust the 10K POT for undistorted output signal.

3. Record the Amplitude of the Input and Output Signals.

| | |
|---|---|
| $V_{in}$ = | SWITCH A DOWN |
| $V_{out}$ = | SWITCH A UP |

4. Calculate the Voltage Gain of the circuit.

$$GAIN = \frac{V_{out}}{V_{in}} = \frac{(\quad)}{(\quad)} =$$

| |
|---|
| GAIN = |

5. Calculate the Theoretical value of the circuit voltage gain using,

$GAIN = gmR_L$

gm = Transconductance (measured in Exp. No. 1)

$R_L$ = Load Resistance = 33K

| |
|---|
| GAIN = ( ) ( ) = |

6. How do the theoretical and actual values for the Gain compare? Note: Differences can be accounted for in the fact that the resistance of the FET itself was never considered.

QUESTIONS:

1. Verify the Output Impedance of the circuit. The Output Impedance should equal $R_L$ = 33K.

   a) Sketch the Circuit you will use to measure output impedance.

   b) Describe the procedure

   c) If a trainer is available, perform the measurement of output impedance.

**Ed-Lab** | FET CHARACTERISTIC | **EXP. NO.** 29 **Page No.** 1

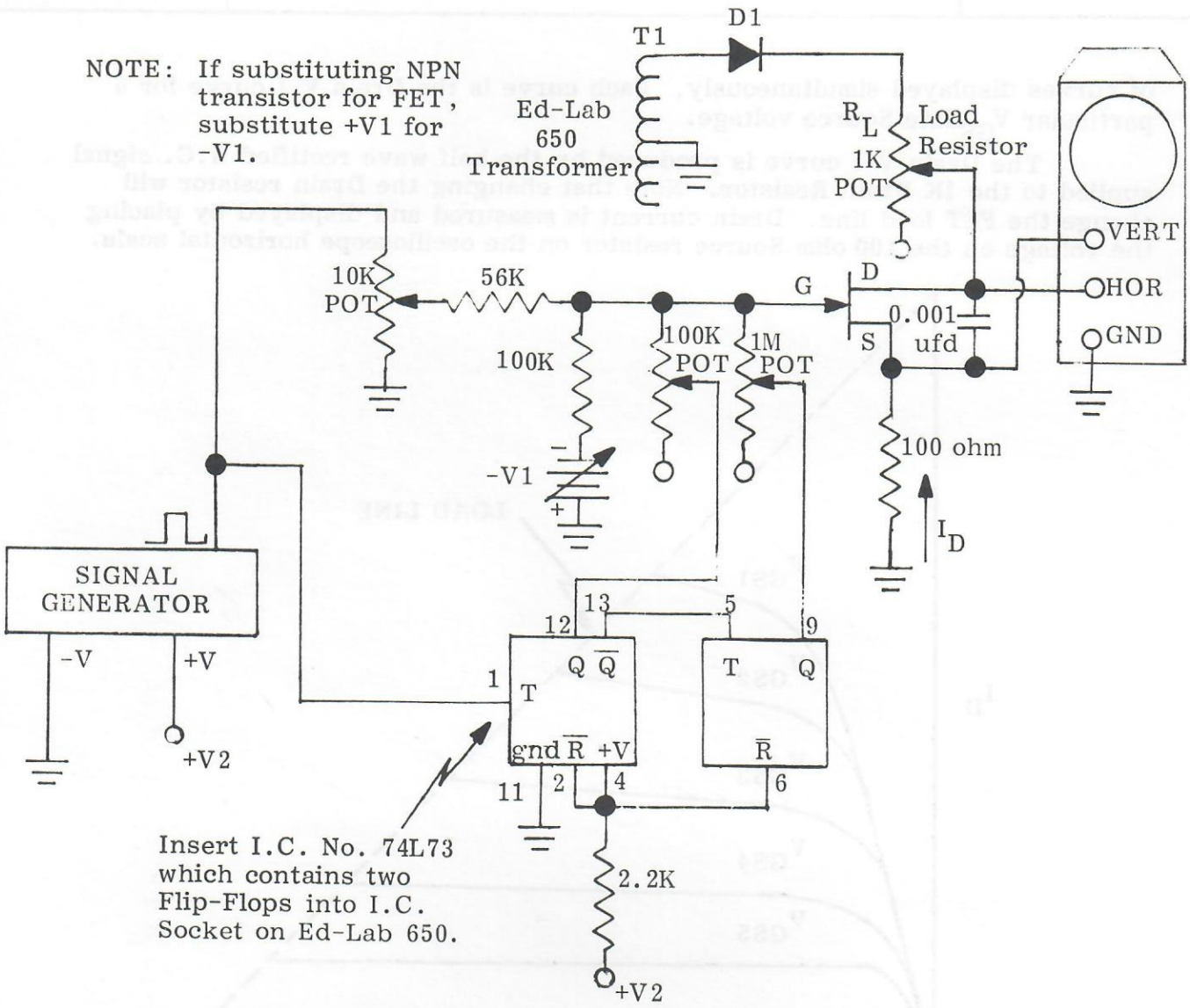



Figure 29-1 Characteristic Curve - FET

Objective:

To generate a family of characteristic curves for the FET circuit and observe the Load Line.

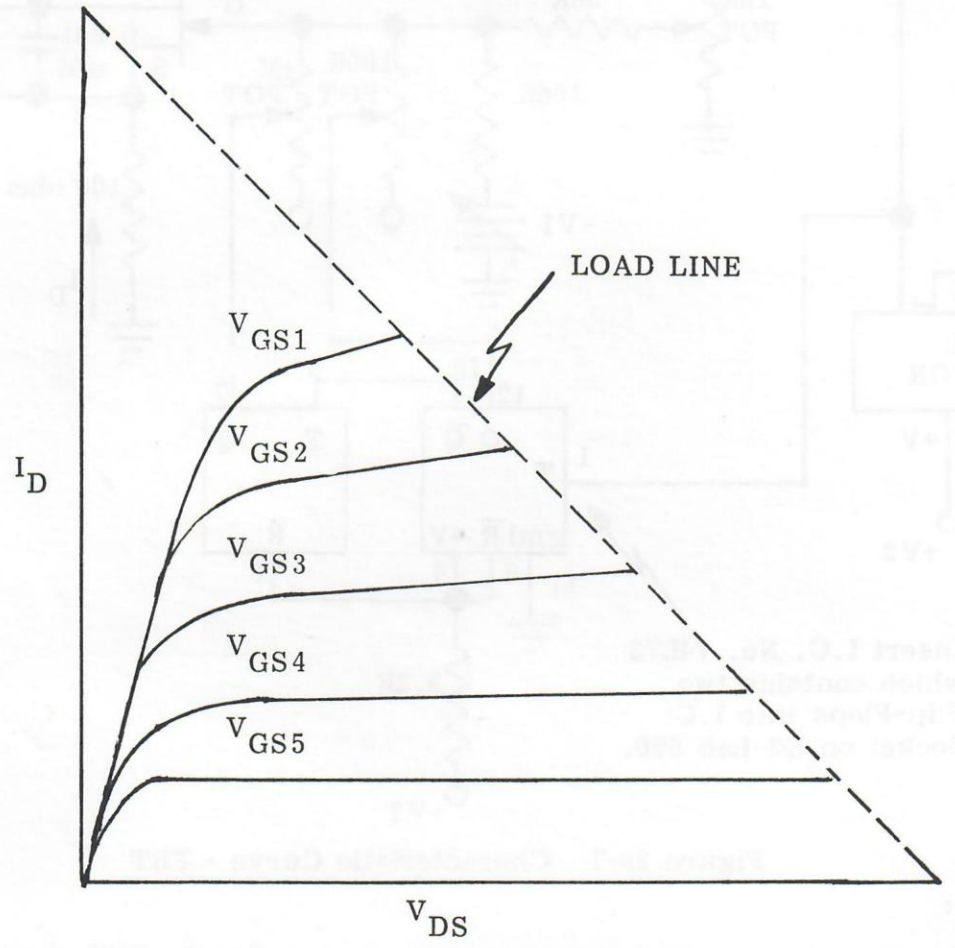
Description:

The FET has a characteristic curve similar to that of the transistor. That is, for each value of $V_{GS}$ the Gate-Source voltage, the Drain Voltage and Current will generate a characteristic plot.

In Figure 29-1the circuit used for displaying a family of characteristic curves is shown. Several different gate voltages are generated via the square wave signal generator driving the 56K resistor and the 74L73 counter driving the 100K and 1 Meg POTS. -V1 is used to adjust the D.C. bias on the Gate voltage. The sequence of Gate voltages is generated at high speed and will produce a family

of curves displayed simultaneously. Each curve is the Drain V-I curve for a particular $V_{GS}$ Gate Source voltage.

The Drain V-I curve is produced by the half wave rectified A.C. signal applied to the 1K Drain Resistor. Note that changing the Drain resistor will change the FET load line. Drain current is measured and displayed by placing the voltage on the 100 ohm Source resistor on the oscilloscope horizontal scale.



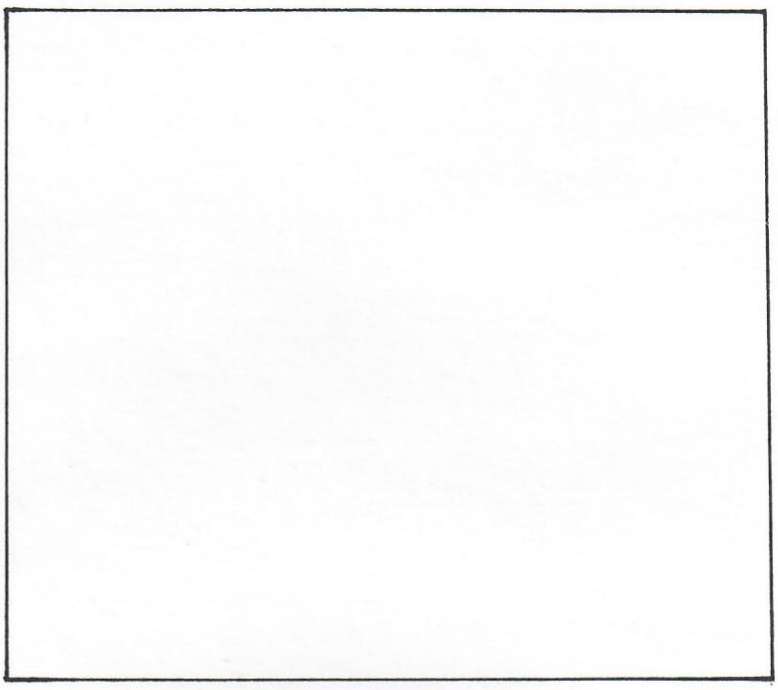

The result will be a trace as pictured in Figure 29-2. Each value of $V_{GS}$ will generate a particular curve which will start at zero volts and zero amps and will follow the curve set by the FET until it reaches the load line. The load line represents the point at which the drain current in limited by the load resistor.

PROCEDURE:

1. Wire the circuit shown in Figure 29-1.
   If a 74L73 I.C. is not available it may simple be eliminated. and the family of characteristics will have only two curves.
2. Set the Oscilloscope for Horizontal Amplifier operation and set the 1K load POT at mid-range.

3. Adjust -V1, 10K, 100K and 1 Meg POTS for a clear set of characteristic curves.

4. Sketch the family of characteristic curves obtained.



5. Change the 1K POT load. Describe the effect on the characteristic curves. Draw the load line for a load resistance of 1000 ohms on the sketch of step 4. This may be done by setting the 1K POT at 1000 ohms (full setting) and observing the family of characteristic curves.

QUESTIONS:

1. What is the ideal bias point (Gate Source Voltage) for a FET linear amplifier as seen on the Characteristic Curve, for a load resistance of 1000 ohms?

| Ed-Lab | FET CONSTANT CURRENT SOURCE | EXP. NO. 30<br>Page No. 1 |
|---|---|---|

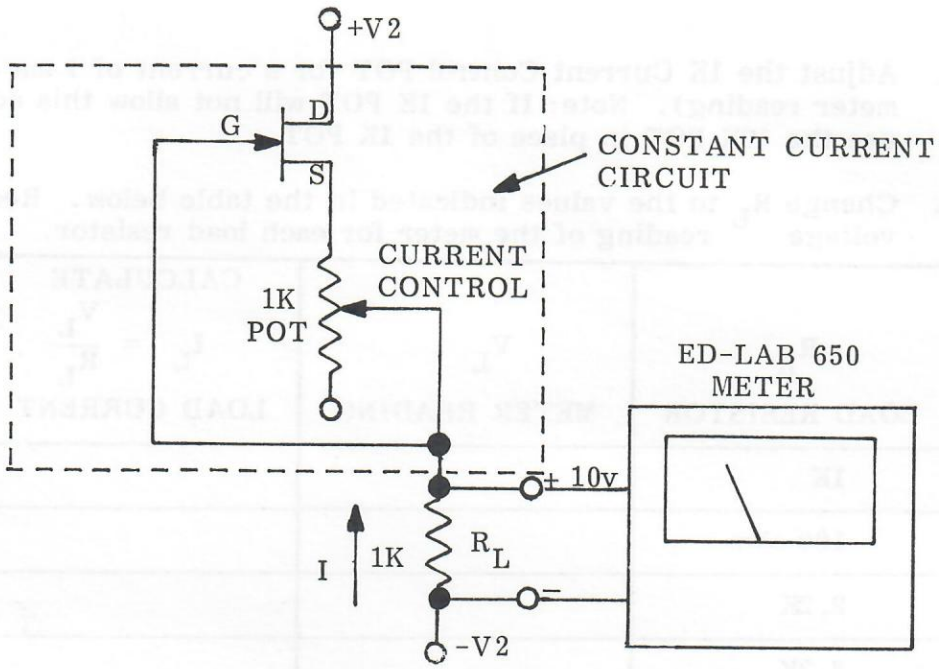



Figure 30-1 FET Constant Current Source

Objective:

You will build and test an FET Constant Current Source to generate a constant current into various loads.

Description:

The circuit of Figure 30-1enclosed in the dotted lines is a constant current generator. Current flowing in the 1K current control POT generates a Gate Source voltage $V_{gs}$. $V_{gs}$ regulates the Drain and Source current. The current output will be set by the resistance of the 1K POT and will be constant for a range of Load Resistances $R_L$.

Current in the load resistor is measured by measuring the voltage generated across the load resistor and using Ohm's Law.

The constant current source will fail for current levels or load resistor values which cause the voltage across the load resistor to exceed the supply voltage minus the $V_{gs}$ generated across the 1K current control POT.

The equation below must be true for the constant current supply to be within operating limits.

$$V_{gs} + I \times R_L < [+V2 - (-V2)]$$

PROCEDURE:

1. Wire the circuit shown in Figure 30-1.

2. Adjust the 1K Current Control POT for a current of 1 ma (1 volt meter reading). Note: If the 1K POT will not allow this adjustment use the 10K POT in place of the 1K POT.

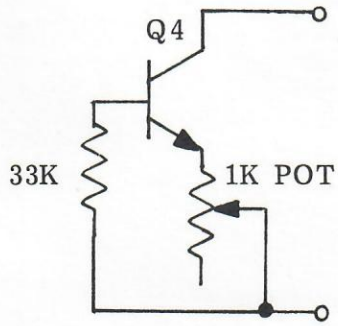
3. Change $R_L$ to the values indicated in the table below. Record the voltage reading of the meter for each load resistor.

| $R_L$ LOAD RESISTOR | $V_L$ METER READING | CALCULATE $I_L = \frac{V_L}{R_L}$ LOAD CURRENT |
|---|---|---|
| 1K | | |
| 100 ohm | | |
| 2.2K | | |
| 3.3K | | |
| 4.7K | | |
| 10K | | |
| 15K | | |
| 22K | | |
| 33K | | |

4. Calculate the LOAD CURRENT and enter the results in the table.
5. Does the circuit operate as a constant current source?
6. For which load resistors does the Constant Current Source fail to generate 1 ma?

QUESTIONS:

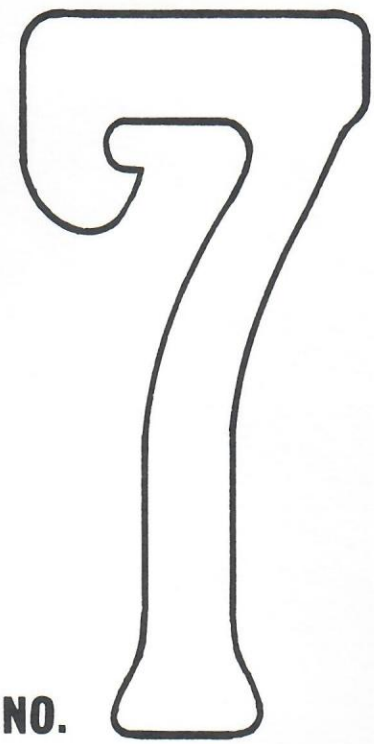
1. Why does the Current Source fail to generate 1 ma for very high load resistors?
2. What is the maximum Output Voltage of this Current Source Circuit?
3. Do you think that a constant current source can be implemented using a similar transistor circuit as shown below?



# Ed-Lab

## *EXPERIMENT MANUAL*

### VACUUM TUBES



**UNIT NO.**

CES INDUSTRIES, INC.

**Ed-Lab** | UNIT NO. 7

# EXPERIMENT MANUAL

# VACUUM TUBES

Copyright © 1974, 1976, 1980, 1981 CES INDUSTRIES, INC.

Name_____

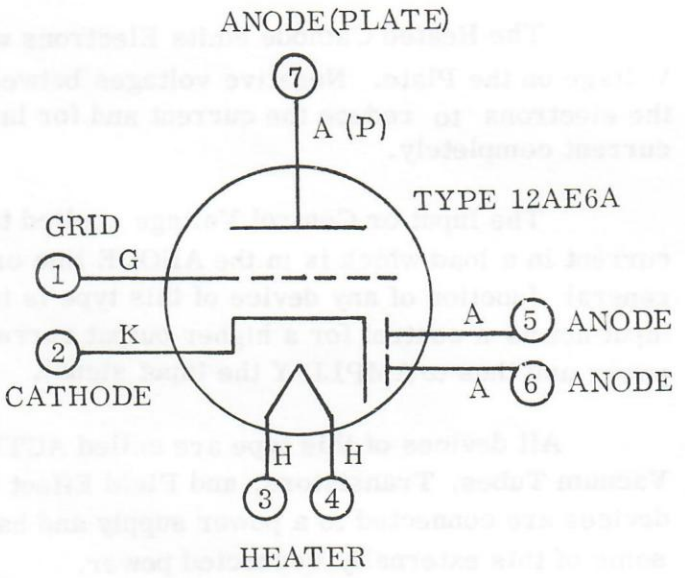
Course_____

TABLE OF CONTENTS

1. Triode D.C. Amplifier
2. Triode Characteristics
3. Plate Characteristic Curve
4. A.C. Triode Grid Bias Amplifier
5. Cathode Bias
6. Cathode Follower Amplifier
7. Diode Rectifier

7 PIN TUBE

TYPE 12AE6A



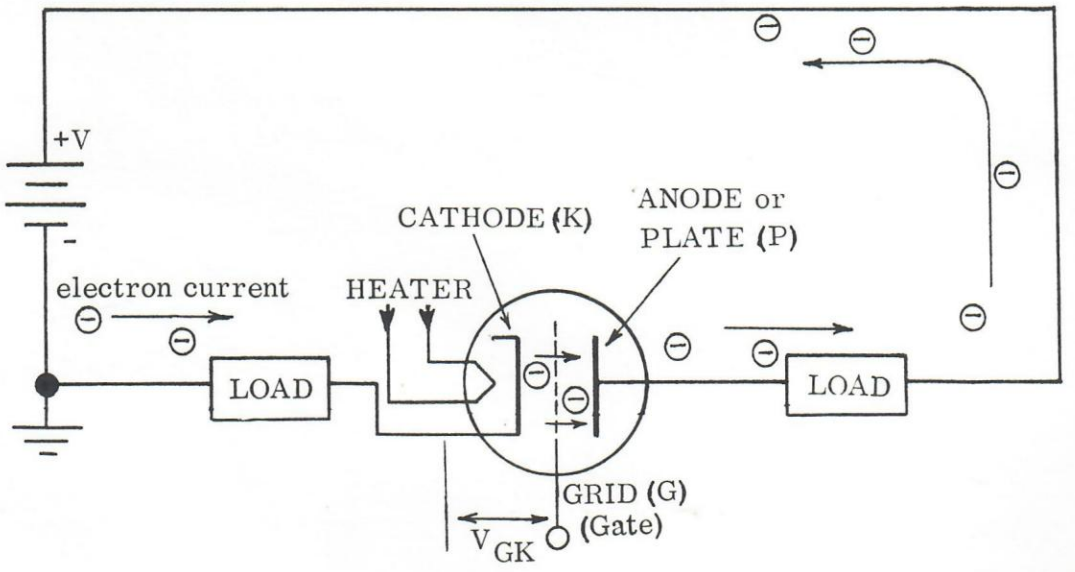

TWIN DIODE / MEDIUM MU TRIODE

General

A 7 PIN-MINIATURE Socket is located on the panel. The Vacuum Tubes used in these experiments are automobile tubes which require a B+ voltage of 12 volts. All power is taken from the built-in Ed-Lab power supply to prevent burning-out of any parts.

DO NOT USE ANY EXTERNAL POWER SUPPLIES.

The Vacuum Tube operates as a Valve or Gate for current similar to certain field effect transistors as shown below:



The Heated Cathode emits Electrons which are attracted to the Positive Voltage on the Plate. Negative voltages between the GRID and CATHODE repel the electrons to reduce the current and for large negative voltages stop the current completely.

The Input or Control Voltage applied to the grid input thus controls the current in a load which is in the ANODE line or in the CATHODE line. The general function of any device of this type is to have a low power or low voltage input act as a control for a higher output current to produce high voltage or high power and thus to AMPLIFY the input signal.

All devices of this type are called ACTIVE ELEMENTS and include Vacuum Tubes, Transistors, and Field Effect Transistors, because these devices are connected to a power supply and have Inputs which act to control some of this externally connected power.

| Ed-Lab | TRIODE D.C. AMPLIFIER | EXP. NO. 1<br>Page No. 1 |
|---|---|---|

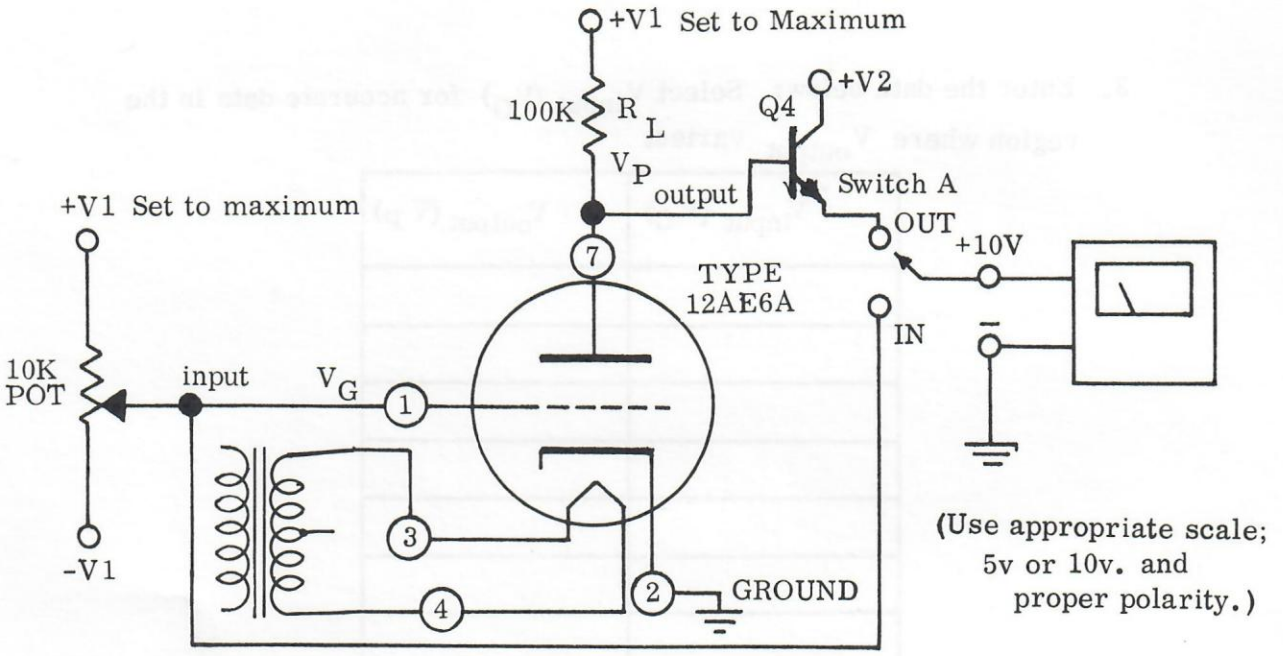


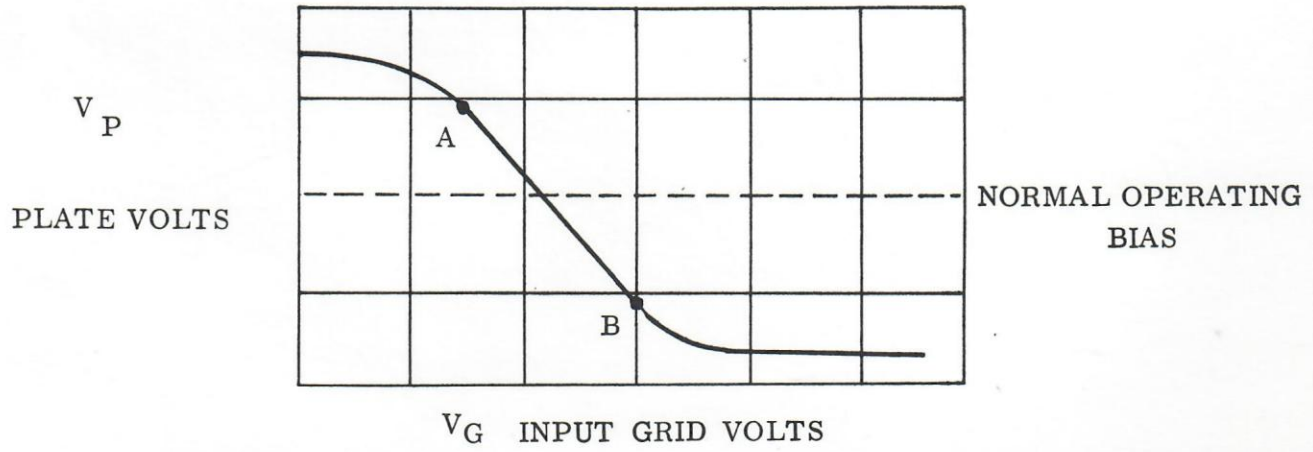

Figure 1-1 Triode Amplifier

Objective:

You will use a Vacuum Tube Triode as a Voltage Amplifier. You will vary the Input GRID Voltage and read the Output PLATE Voltage. You will also calculate the Amplifier Gain and Vacuum Tube Gain factors.

Procedure:

1. Wire the circuit in Figure 2-1 above.
2. Vary the Input Voltage using the 10K POT and read the output and input voltages. Use the meter to measure these voltages and enter the data in the table on the following page. The Output Voltage versus Input Voltage is then plotted. The curve should appear as shown below:

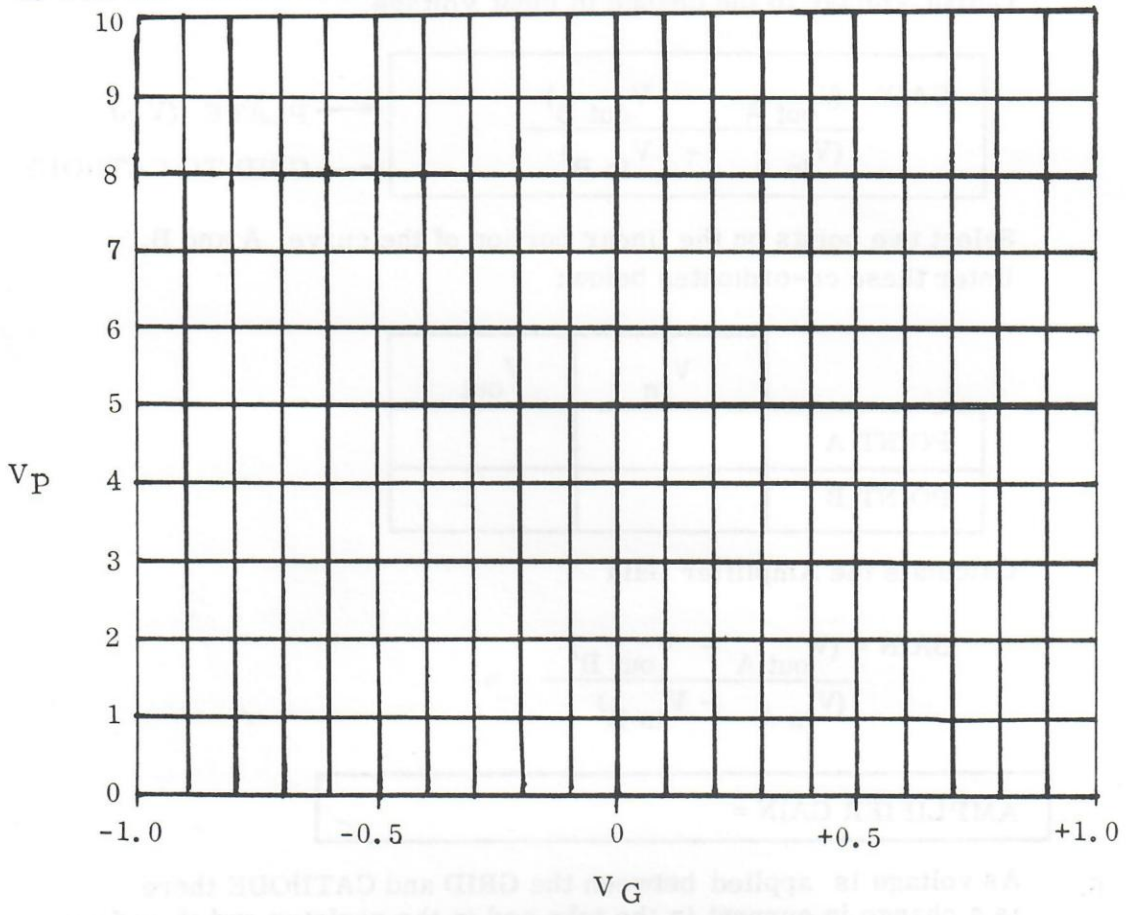


3. Enter the data below: Select $V_{input}$ ($V_G$) for accurate data in the region where $V_{output}$ varies.

| $V_{input}$ ($V_G$) | $V_{output}$ ($V_P$) |
|---|---|
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |

**Ed-Lab** | TRIODE D.C. AMPLIFIER | **EXP. NO.** 1 **Page No.** 3

4. Plot the results on the graph below:




The linear operating range is from point A to point B on the curve shown on Page 1. What is this linear operating range from your curve plotted above?

| | | |
|---|---|---|
| LINEAR RANGE | $V_{input}$ | FROM_____volts to _____volts |
| | $V_{output}$ | FROM_____volts to _____volts |

Select a normal operating middle BIAS point:

| MID-BIAS POINT |
|---|
| $V_{plate\ voltage}$ = |
| $V_{grid\ voltage}$ = |

5. The Amplifier Gain in the linear range is the ratio of the change in Output Voltage to the change in Input Voltage.

$$\text{GAIN} = \frac{(V_{\text{out A}} - V_{\text{out B}})}{(V_{\text{in A}} - V_{\text{in B}})}$$

← PLATE ($V_P$)

← GRID TO CATHODE ($V_G$)

Select two points on the linear portion of the curve, A and B. Enter these co-ordinates below:

| | $V_{in}$ | $V_{out}$ |
|---|---|---|
| POINT A | | |
| POINT B | | |

Calculate the Amplifier Gain

$$\text{GAIN} = \frac{(V_{\text{out A}} - V_{\text{out B}})}{(V_{\text{in A}} - V_{\text{in B}})} =$$

| AMPLIFIER GAIN = |
|---|

6. As voltage is applied between the GRID and CATHODE there is a change in current in the tube and in the resistor and therefore in the output voltage. Three factors describe the operation of the tube. These are:

$\mu$ = AMPLIFICATION FACTOR

(GREEK LETTER MU.)

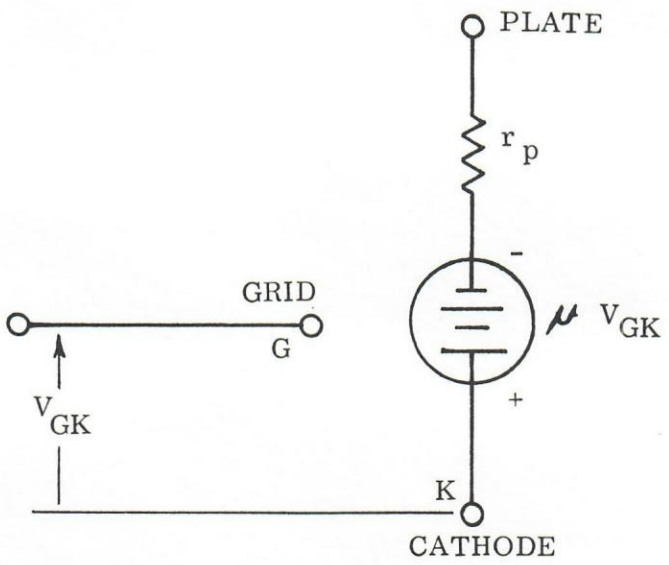
$r_p$ = PLATE RESISTANCE

$g_m$ = TRANSCONDUCTANCE
(also Mutual Conductance)

These three factors are related as:

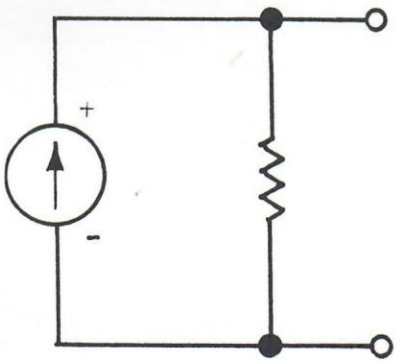
$$g_m = \frac{\mu}{r_p}$$

The "equivalent circuit" for a vacuum tube is:




This is then a "Thevenin's Equivalent Circuit" with a voltage source corresponding to the Amplification Factor $\mu$ times the input voltage, $V_{GK}$ and the series "Plate Resistance". The equivalent "Norton's Current Circuit" is given below:

$$I = \frac{V_{GK}}{r_p} = g_m V_{GK}$$




The tube characteristics are reviewed and measured in the following experiment.

| Ed-Lab | TRIODE CHARACTERISTICS | EXP. NO. 2<br>Page No. 1 |
|---|---|---|

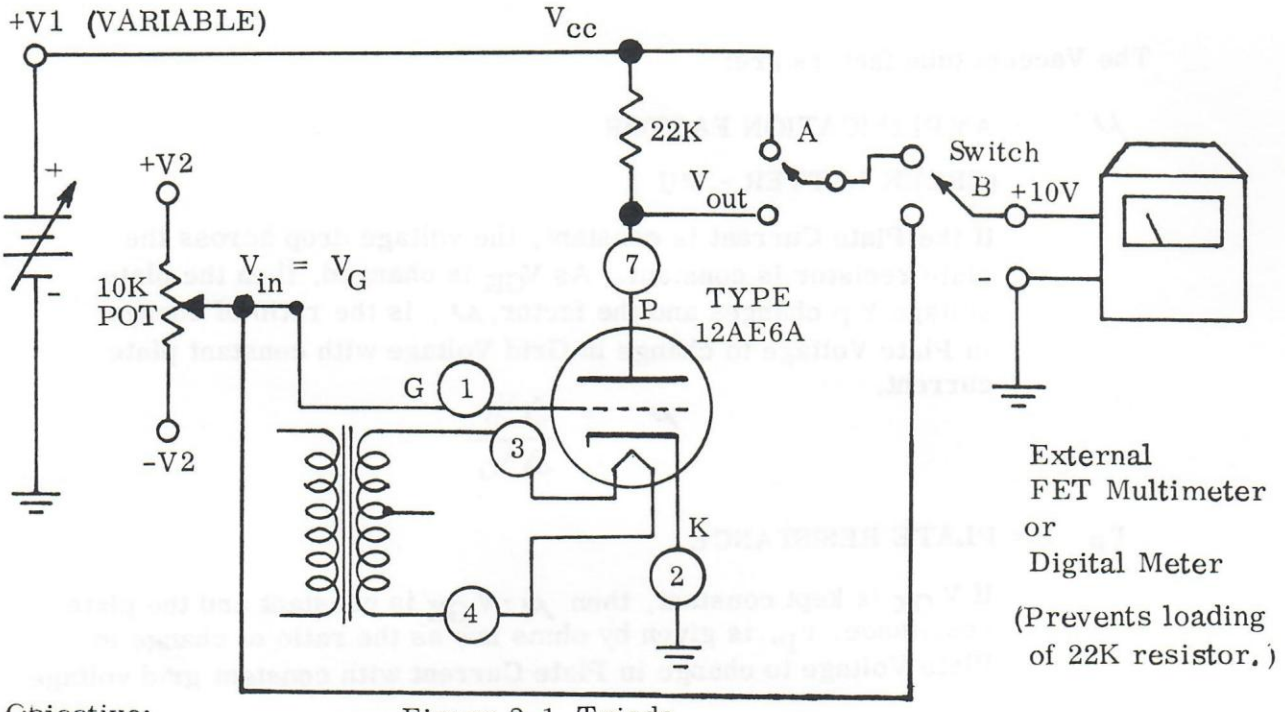



Figure 2-1 Triode

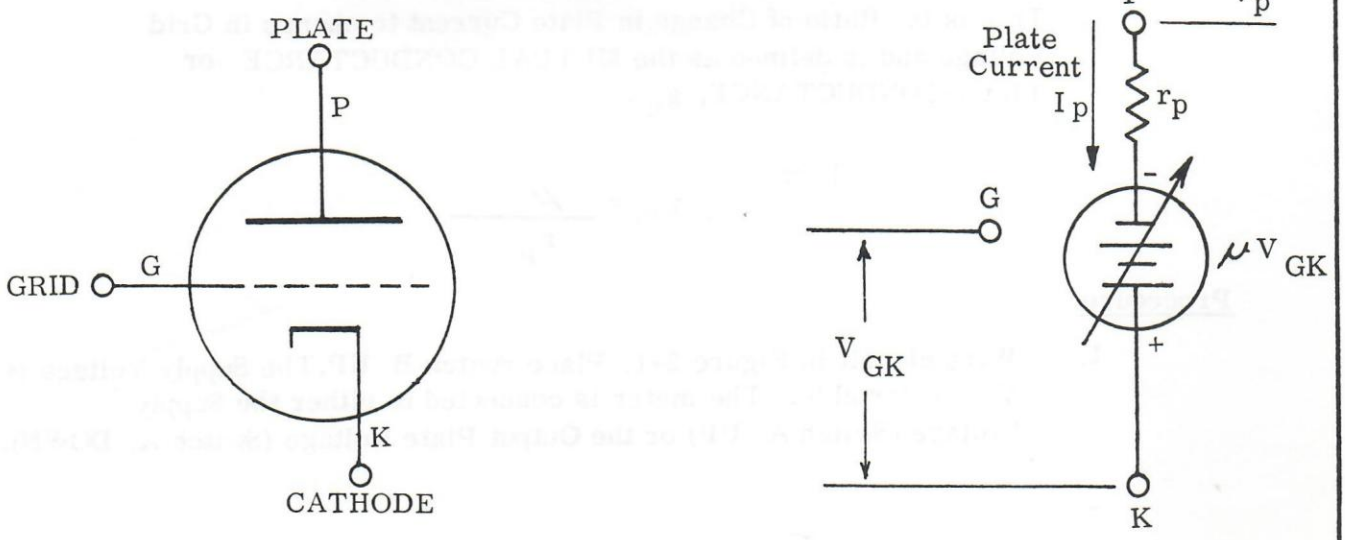
Objective:

You will review the CHARACTERISTICS and FACTORS associated with the TRIODE VACUUM TUBE AMPLIFIER .

Description:

When the Input Voltage ($V_G$) applied to the GRID is changed, there is a change in PLATE Current, resulting in a change in voltage across the Load Resistor and therefore a change in the Output Voltage.

The EQUIVALENT Circuit for the TRIODE is shown below:



The Vacuum tube factors are:

$\mu$ = AMPLIFICATION FACTOR

(GREEK LETTER - MU )

If the Plate Current is constant, the voltage drop across the plate resistor is constant. As $V_{GK}$ is changed, then the plate voltage $V_p$ changes and the factor, $\mu$, is the ratio of change in Plate Voltage to change in Grid Voltage with constant plate current.

$$\mu = \frac{\Delta V_p}{\Delta V_G}$$

$r_p$ = PLATE RESISTANCE

If $V_{GK}$ is kept constant, then $\mu \cdot V_{GK}$ is constant and the plate resistance, $r_p$, is given by ohms law as the ratio of change in Plate Voltage to change in Plate Current with constant grid voltage

$$r_p = \frac{\Delta V_p}{\Delta I_p} \qquad V_{GK} = \text{constant}$$

The ratio of $\mu$ to $r_p$ is the Norton Current Source and is

$$\frac{\mu}{r_p} = \frac{\Delta V_p}{\Delta V_G} \cdot \frac{\Delta I_p}{\Delta V_p} = \frac{\Delta I_p}{\Delta V_G}$$

This is the Ratio of Change in Plate Current to change in Grid Voltage and is defined as the MUTUAL CONDUCTANCE or TRANSCONDUCTANCE, $g_m$.

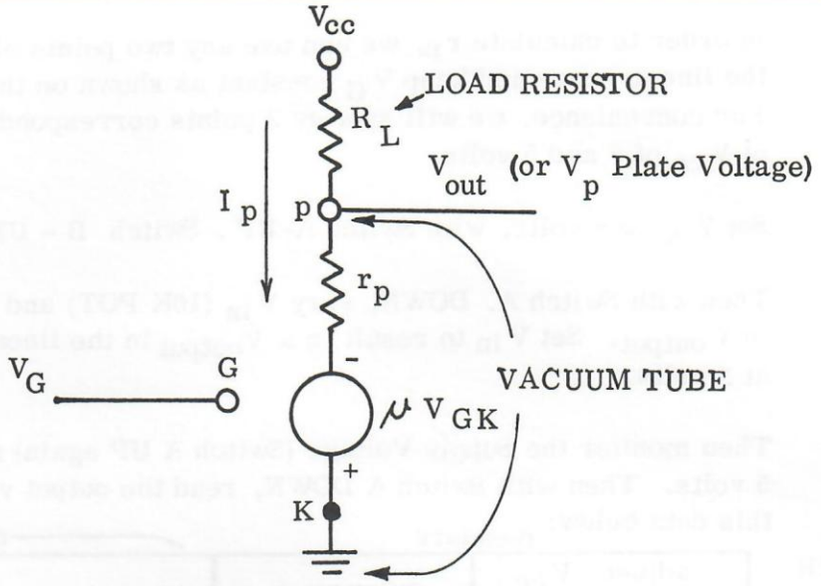
Then:

$$g_m = \frac{\mu}{r_p}$$

Procedure:

1. Wire circuit in Figure 2-1. Place Switch B UP. The Supply Voltage ($+V_1$) $V_{cc}$ is variable. The meter is connected to either the Supply Voltage (Switch A, UP) or the Output Plate Voltage (Switch A, DOWN).

The total equivalent circuit is shown on the following page:




2. The Amplifier Gain, K, can be calculated for any amplifier if we know the TUBE $\mu$ and $r_p$ and the CIRCUIT $R_L$:

$$K = \frac{\Delta V_{out}}{\Delta V_{GK}} = \mu \left( \frac{R_L}{R_L + r_p} \right)$$

3. TO MEASURE & CALCULATE $r_p$:

If $V_{GK}$ is constant, then $\mu V_{GK}$ is constant, and by Ohm's Law:

$$r_p = \frac{\Delta V_{out}}{\Delta I_p}$$

$\Delta$ = (delta or difference) change from two sets of readings.

Two different readings are obtained by varying $V_{cc}$.

where $I_p = \frac{(V_{cc} - V_{out})}{R_L}$

4. TO MEASURE & CALCULATE $\mu$:

Similarly, if there is a change in $V_{GK}$($\Delta V_G$), then the change in Output Voltage ( $\Delta V_{out}$) is given by the voltage divider equation:

$$\Delta V_{out} = \mu \Delta V_G \left( \frac{R_L}{R_L + r_p} \right)$$

Rather than attemtping to keep $I_p$ constant we can measure the gain and $\mu$ by varying $V_G$ and measuring $V_{out}$.

5. In order to calculate $r_p$, we can use any two points of operation in the linear range and keep $V_G$ constant as shown on the previous page. For convenience, we will specify 2 points corresponding to settings of $V_{cc}$ of 8 and 5 volts.

Set $V_{cc}$ to 8 volts, with Switch A-UP , Switch B - UP.

Then with Switch A, DOWN, vary $V_{in}$ (10K POT) and observe the variation in $V_{output}$. Set $V_{in}$ to result in a $V_{output}$ in the linear range of operation at 3 volts.

Then monitor the Supply Voltage (Switch A UP again) and reduce $V_{cc}$ to 5 volts. Then with Switch A DOWN, read the output voltage, $V_p$, and enter this data below:

| | measure | | calculate | |
|---|---|---|---|---|
| SWITCH A - UP | adjust $V_{cc}$ SWITCH A UP | SWITCH A DOWN | $V_L = V_{cc} - V_p$ | $I_p = \frac{V_L}{22K}$ |
| POINT 1 | $V_{cc\,1}$ = 8 volts | (vary $V_G$ for) $V_p$ = 3 volts | $V_{L\,1}$ = 5 volts | $I_{p1} = \frac{5}{22K}$ = 227 $\mu$a |
| POINT 2 | $V_{cc\,2}$ = 5 volts | (same $V_G$) $V_p$ = | $V_{L\,2}$ = | $I_{p\,2}$ = |

6. Calculate the voltage across the load resistor($V_L$) and the load or plate current ($I_p$) and enter the results in the table for Point 2.

Example for the Point 1:

for $V_p$ = 6 volts

and $V_{cc}$ = 8 volts

and $R_L$ = 22K

$V_L = (V_{cc} - V_p) = (8 - 6) = 2$

$I_p = \frac{V_L}{R_L} = \frac{2}{22K}$ = 90 microamperes

7. Using the calculated data, then calculate

$$r_p = \frac{\triangle V_p}{\triangle I_p} = \frac{(V_{p\,1} - V_{p2})}{(I_{p\,1} - I_{p2})} \quad \text{(CONSTANT } V_G\text{)}$$

$r_p$ = ohms

8. We can now calculate $\mu$ . Two readings are now taken with a fixed $V_{cc}$ and a variable $V_G$ input voltage. Place SWITCH B DOWN. Set $V_{cc}$ at maximum. You will use two Input ($V_G$) voltages and read the two output voltages ($V_p$ or $V_{output}$) in the linear range of operation. Enter the data below:

| | SWITCH A DOWN ( SWITCH B UP ) $V_{output}$ (plate volts) | SWITCH B DOWN Read $V_G$ (input volts) Switch to negative meter if necessary |
|---|---|---|
| POINT 1 | Vary input POT for: $V_{out\,1}$ = 5 volts | $V_{G\,1}$ = |
| POINT 2 | Vary input POT for: $V_{out\,2}$ = 2 volts | $V_{G\,2}$ = |

9. Calculate $\mu$ as described on page 3 from:

$$\Delta V_{out} = \mu \, \Delta V_{in} \cdot \left( \frac{R_L}{R_L + r_p} \right)$$

$$\text{or} \quad \mu = \frac{\Delta V_{out}}{\Delta V_{in}} \cdot \left( \frac{R_L + r_p}{R_L} \right)$$

$$\mu = \frac{(V_{out\,1} - V_{out\,2})}{(V_{G1} - V_{G2})} \cdot \left( \frac{R_L + r_p}{R_L} \right)$$

Substitute the values for $V_{G1}$, $V_{G2}$, $V_{out1}$, $V_{out2}$, the calculated $R_p$, and $R_L$ = 22K . Then calculate $\mu$.

| $\mu$ = |
|---|

10. Given $\mu$ and $r_p$ as measured, (for a tube) we can now calculate the gain of an AMPLIFIER using any load. The gain from the equation in paragraph 2 is:

$$K = \mu \left( \frac{R_L}{R_L + r_p} \right)$$

Using the $\mu$ and $r_p$ as measured and calculated, and an $R_L$ = 22K calculate the Amplifier Gain:

K = _____

11. How does this calculated gain, K, compare to the measured amplifier gain from Experiment No. 1 which used a load resistance of 100K?

12. How would you select $R_L$ for maximum voltage gain?

13. What $R_L$ would you select for maximum power gain (recall the
14. Maximum Power Transfer experiment)?

What is the gain for an $R_L$ of 200K?

15. What is the maximum voltage gain you could obtain using any load with this TRIODE?

16. Calculate the Transconductance, $g_m$. The "dimension" is the "inverse" of OHM and is called a MHO.

$$g_m = \frac{\mu}{r_p} \text{ MHOs}$$

**Ed-Lab** | PLATE CHARACTERISTIC CURVE | **EXP. NO.** 3 **Page No.** 1

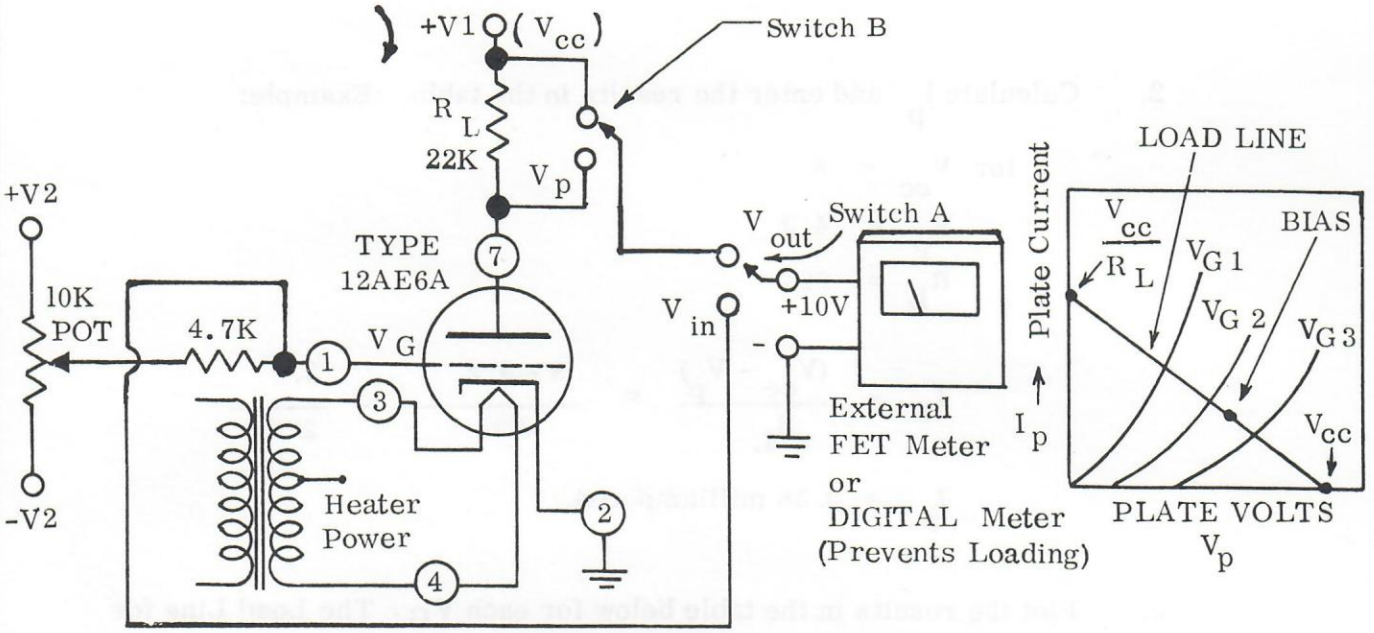



Figure 3 - 1

Objective:

You will measure the points on the curve for $V_p$ versus $I_p$ for constant $V_G$. A series of these curves are shown above and is called the AVERAGE or STATIC PLATE CHARACTERISTICS. The LOAD LINE is plotted to show DYNAMIC operation for varying or A.C. signals.

Procedure:

1. Wire the circuit. For constant $V_G$, vary $V_{cc}$ ($+V_1$) and read $V_{cc}$ and $V_p$. Then calculate $I_p = \frac{(V_{cc} - V_p)}{R_L}$. Enter all data in the table below:

Switch B UP
Switch A UP

| | Switch B Down: SET $V_G$ AND READ $V_P$ | | | | Calculate $I_p$ | | | |
|---|---|---|---|---|---|---|---|---|
| Set $V_{cc}$ (Vary +V1) | $V_G = 0$ Switch A Down A | $V_G = -.25$ B | $V_G = -.5$ C | $V_G = -.75$ D | $I_p$ A | $I_p$ B | $I_p$ C | $I_p$ D |
| 8 | Switch A UP $V_p =$ | | | | | | | |
| 6 | $V_p =$ | | | | | | | |
| 4 | $V_p =$ | | | | | | | |
| 2 | $V_p =$ | | | | | | | |
| 0 | $V_p =$ | | | | | | | |

| Ed-Lab | PLATE CHARACTERISTIC CURVE | EXP. NO. 3<br>Page No. 2 |
|---|---|---|

2. Calculate $I_p$ and enter the results in the table. Example:

for $V_{cc} = 8$

$V_p = 4.2$

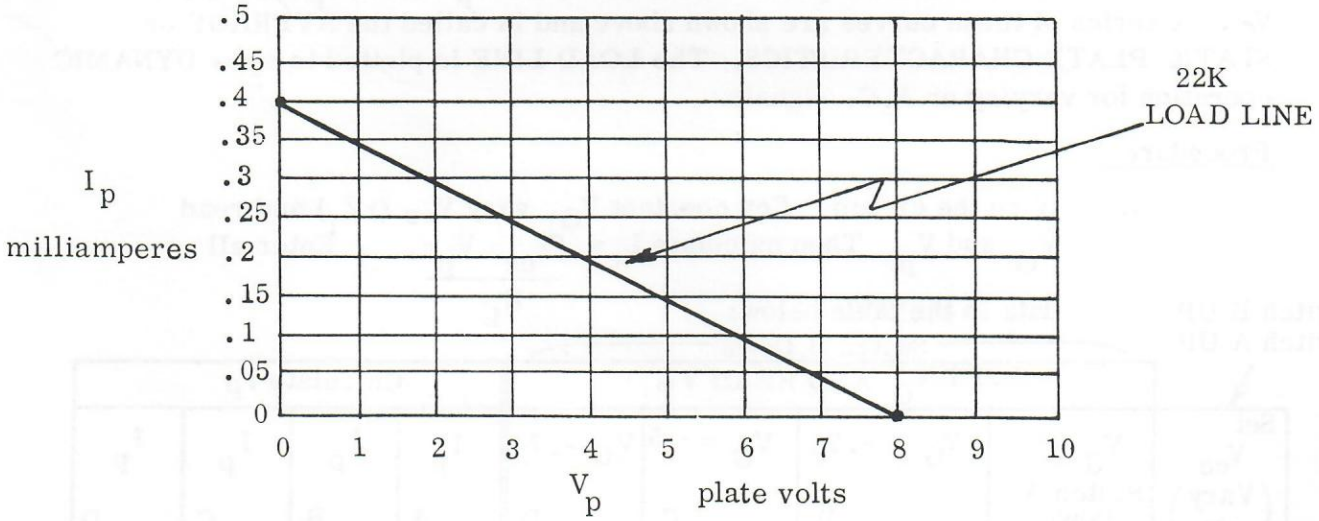
$R_L = 22K$

$$I_p = \frac{(V_{cc} - V_p)}{R_L} = \frac{8 - 4.2}{22K} = \frac{3.8}{22K}$$

$I_p = 0.38$ milliamperes.

3. Plot the results in the table below for each $V_G$. The Load Line for $R_L = 22K$ is shown on the graph. When $V_p = V_{cc} = 8$ volts; $I_p = 0$.

When $V_p = 0$ $I_p = \frac{V_{cc}}{R_L} = \frac{8}{22K} = 0.36$ ma.




4. Draw the Load Line for an $R_L = 33K$ Load.

5. Select a Plate BIAS operating point on the $R_L = 33K$ load line for an A.C. Amplifier.

What bias voltage would you use on the grid to obtain this plate BIAS?

**Ed-Lab** | A.C. TRIODE GRID BIAS AMPLIFIER | **EXP. NO.** 4 **Page No.** 1

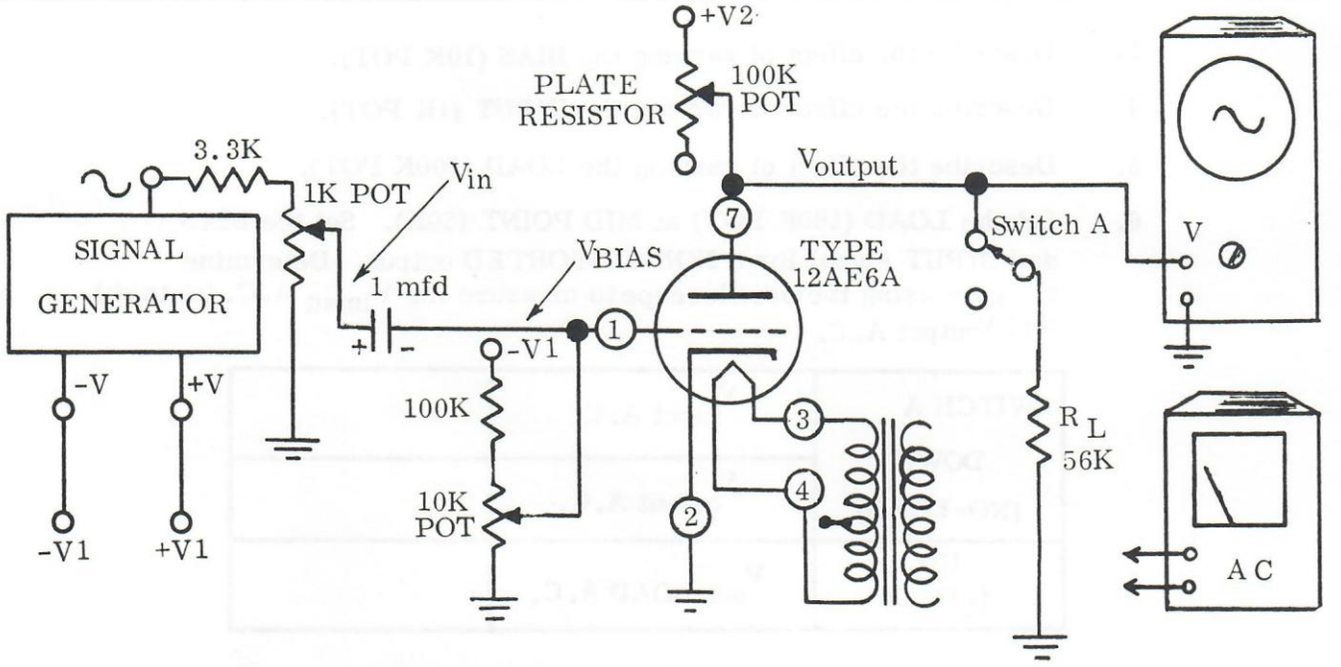



Figure 4-1

Objective:

You will use an external BIAS voltage on the GRID to set an operating Bias Point. An A.C. signal is provided as an input and the output is observed and the gain is measured for different load resistors.

Procedure:

1. Wire the circuit in Figure 4 - 1. Keep Switch A DOWN providing no load. Set the 100K POT Load at approximately 1/2 way (50K).
2. Vary the BIAS VOLTAGE (10K POT) and the INPUT VOLTAGE ($V_{in}$) (1K POT) and observe the results on the oscilloscope.

Plot the waveforms below:

DISTORTED WAVEFORM

UN-DISTORTED WAVEFORM

| Ed-Lab | A.C. TRIODE GRID BIAS AMPLIFIER | EXP. NO. 4<br>Page No. 2 |
|---|---|---|

3. Describe the effect of varying the BIAS (10K POT).

4. Describe the effect of varying the INPUT (1K POT).

5. Describe the effect of varying the LOAD (100K POT).

6. Set the LOAD (100K POT) at MID POINT (50K). Set the BIAS and INPUT signal for a NON-DISTORTED output. Determine the gain using the Oscilloscope to measure the $V_{input}$ A.C. (pk to pk) and $V_{output}$ A.C.

| | | |
|---|---|---|
| SWITCH A DOWN (NO-LOAD) | $V_{input}$ A.C. | |
| | $V_{output}$ A.C. | |
| UP (LOAD) | $V_{out}$ LOAD A.C. | |

Calculate the MEASURED GAIN from above (with NO-LOAD.)

$$\text{MEASURED GAIN} = \frac{V_{out}\ \text{A.C.}}{V_{in}\ \text{A.C.}} =$$

Calculate the THEORETICAL GAIN from:

$$\text{THEORETICAL GAIN} = \mu \left( \frac{R_L}{R_L + r_p} \right) =$$

Use $\mu$ and $r_p$ from the previous experiments.

Use $R_L$ = 50K.

7. How do the measured and Theoretical gain compare? Why is there a difference?

| Ed-Lab | CATHODE BIAS | EXP. NO. 5<br>Page No. 1 |
|---|---|---|

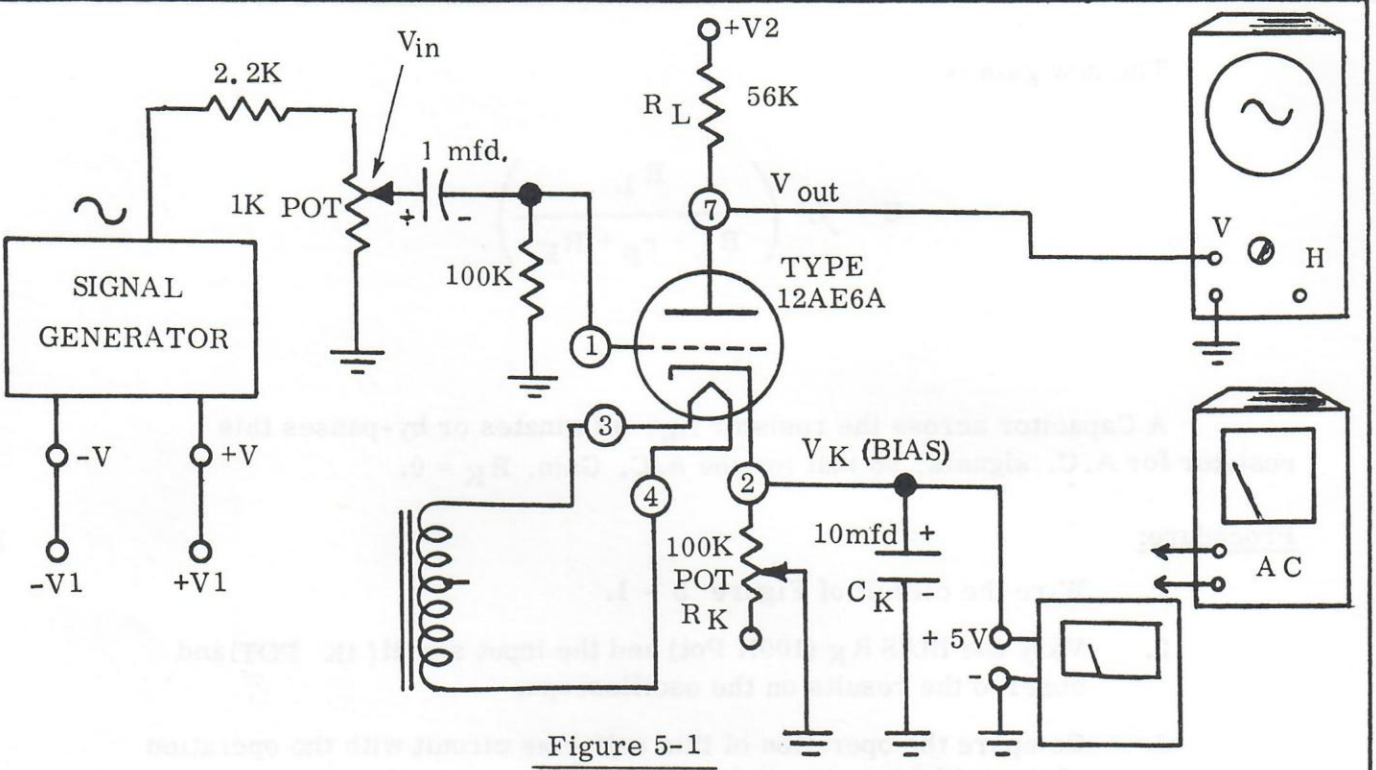



Figure 5-1

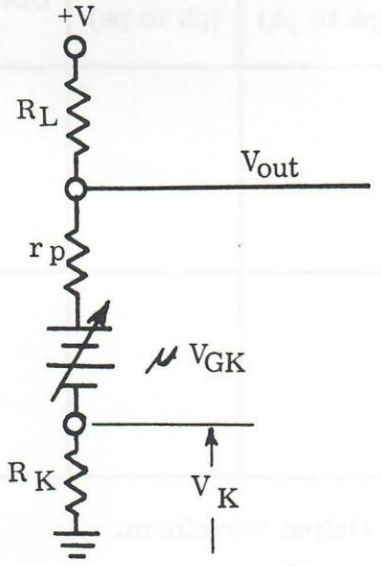
Objective:

You will use a CATHODE RESISTOR to provide a SELF-BIAS voltage. A BYPASS CAPACITOR is used to avoid DEGRADATION or reduction in GAIN.

Description:

With a Cathode Resistor ($R_K$), the Plate Current $I_p$ in the Cathode produces a positive cathode voltage ($V_K$) of $R_K I_p$ which has the same effect as a negative GRID BIAS.

The equivalent circuit is:



The new gain is:

$$G = \mu \left( \frac{R_L}{R_L + r_p + R_K} \right)$$

A Capacitor across the resistor $R_K$, eliminates or by-passes this resistor for A.C. signals, so that for the A.C. Gain, $R_K = 0$.

Procedure:

1. Wire the circuit of Figure 5 - 1.
2. Vary the BIAS $R_K$ (100K Pot) and the input signal (1K POT) and observe the results on the oscilloscope.
3. Compare the operation of this self-bias circuit with the operation of the grid-bias circuit from the previous experiment.
4. Adjust the Input Signal (1K POT) and Bias Point (100K Pot) for a non-distorted signal output as viewed on the oscilloscope. Use the Oscilloscope to measure the input and output voltages with and without the By-Pass Capacitor. Enter the data below:

| | MEASURE | | | CALCULATE |
|---|---|---|---|---|
| | $V_{in}$ A.C. (pk to pk) | $V_{out}$ A.C. (pk to pk) | $V_K$ BIAS D.C. | GAIN = $\frac{V_{out}\ A.C.}{Vin\ A.C.}$ |
| Test 1 | | | | |
| Test 2 Remove bypass CAPACITOR | | | | |

5. Sketch the Output Waveform.
6. Compare both Gains. Compare with the Gain in Experiment No. 4.
7. Compare both Bias Voltages.

| Ed-Lab | CATHODE FOLLOWER AMPLIFIER | EXP. NO. 6<br>Page No. 1 |
|---|---|---|

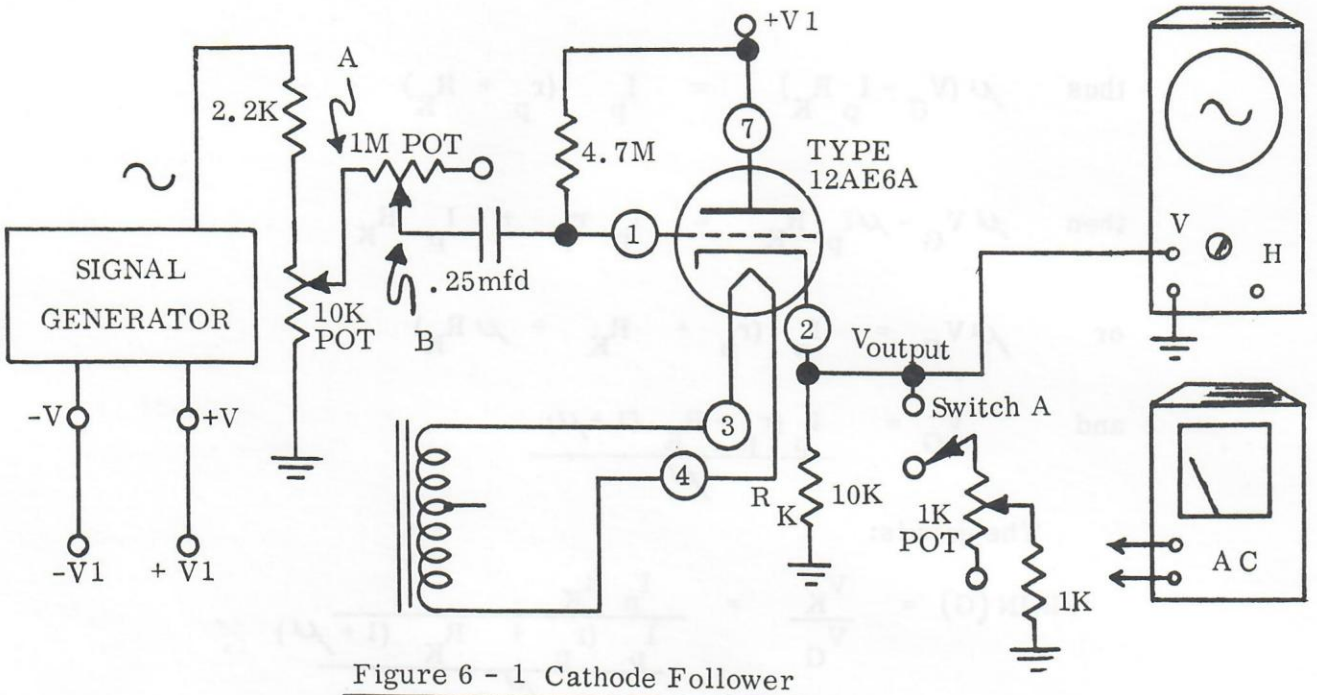


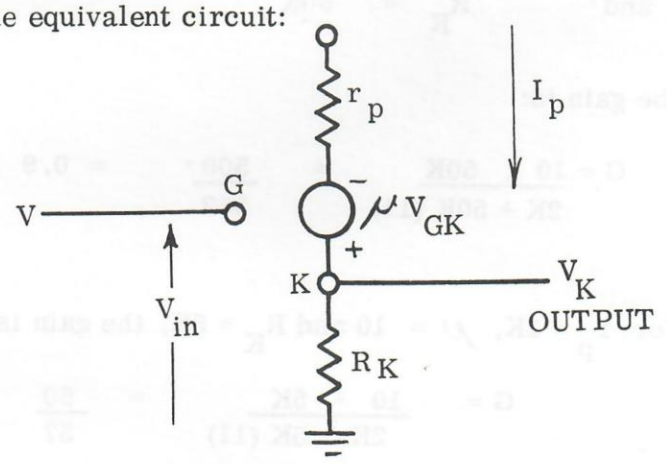

Figure 6 - 1 Cathode Follower

Objective:

You will review the operation of the CATHODE FOLLOWER circuit and measure the Voltage Gain, Input Impedance, Output Impedance, and Power Gain.

Description:

Consider the equivalent circuit:




Using Ohm's Law:

$$\mu V_{GK} = I_p (r_p + R_K)$$

but $$V_K = I_p R_K$$

and $$V_{GK} = V_G - V_K = V_G - I_p R_K$$

thus $\mu (V_G - I_p R_K) = I_p (r_p + R_K)$

then $\mu V_G - \mu I_p R_K = I_p r_p + I_p R_K$

or $\mu V_G = I_p (r_p + R_K + \mu R_K)$

and $V_G = \dfrac{I_p (r_p + R_K (1 + \mu)}{\mu}$

The gain is:

$$\text{GAIN}(G) = \frac{V_K}{V_G} = \frac{I_p R_K}{\dfrac{I_p (r_p + R_K (1 + \mu)}{\mu}}$$

$$G = \frac{\mu R_K}{r_p + R_K (1 + \mu)}$$

Consider $r_p = 2000 \text{ ohms} = 2K$

and $\mu = 10$

and $R_K = 50K$

The gain is:

$$G = \frac{10 \cdot 50K}{2K + 50K (11)} = \frac{500}{552} = 0.9$$

For $r_p = 2K$, $\mu = 10$ and $R_K = 5K$, the gain is:

$$G = \frac{10 \cdot 5K}{2K + 5K (11)} = \frac{50}{57} = .88$$

Note that the Gain is a little less than 1 in both cases and does not vary greatly with a much lower $R_K$, which is in parallel with the Load Resistor. This shows that as a load is applied, there is very little "loading" which indicates a "low output impedance".

| Ed-Lab | CATHODE FOLLOWER AMPLIFIER | EXP. NO. 6<br>Page No. 3 |
|---|---|---|

Procedure:

1. Wire the circuit in Figure 6 - 1. Keep Switch A DOWN, so that there is NO LOAD.

   Adjust the 1M POT to the zero setting for "0" input resistance.

2. Observe the signal on the oscilloscope. Set various A.C. input voltages (10K POT) and read the corresponding output A.C. voltages. Enter the data below:

| $V_{in}$ A.C. | $V_{out}$ A.C. |
|---|---|
| | |
| | |
| | |
| | |
| | |

3. Calculate this measured gain using any set of readings

$$\text{GAIN} = \frac{V_{out} \text{ A.C.}}{V_{in} \text{ A.C.}} =$$

| GAIN = |
|---|

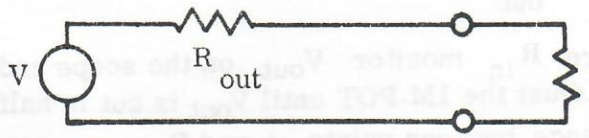
4. Measure the Output Impedance.

   Set one $V_{in}$ A.C. with a corresponding $V_{out}$ A.C. monitored on the Oscilloscope.

   Record this output level:

| $V_{out}$ A.C. = |
|---|

5. Place Switch A UP to insert a LOAD consisting of the 1K POT. Adjust the 1K POT until the output reads <u>half</u> of the $V_{out}$ A.C. setting in paragraph 4 above. Using Thevenin's Theorem and Voltage Divider theory, this load is equal to the output impedance.

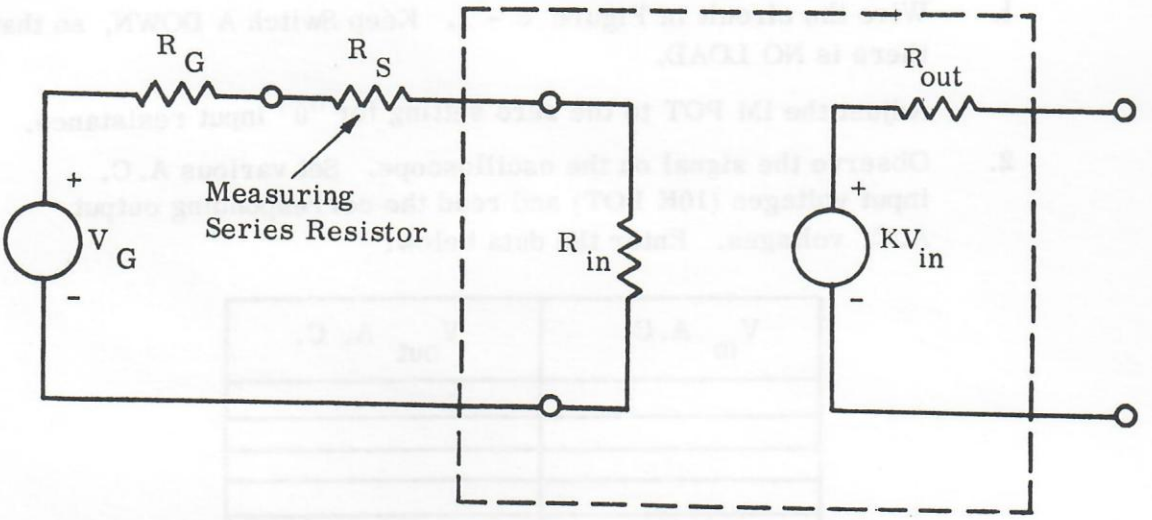



(1K plus POT SETTING)

Estimate the resistance setting of the 1K Potentiometer.

| $R_{output}$ = |
|---|

6. Measure the Input Impedance




A general amplifier is shown above. When an input voltage generator, with an internal resistance, $R_G$, is connected to the amplifier input, the voltage divider equation results in a reduction in voltage due to the amplifier input resistance $R_{in}$

$$V_{in} = V_G \left( \frac{R_{in}}{R_{in} + R_G} \right)$$

It is generally desirable to have a large $R_{in}$ and a small $R_G$ to provide a maximum voltage. With Switch A UP, another series resistor is inserted, $R_S$, and the new voltage is:

$$V_{in} = V_G \left( \frac{R_{in}}{R_{in} + R_G + R_S} \right)$$

We can measure the input resistance, $R_{in}$, by adding a series resistance and noting that when $V_{in}$ is decreased to half, the total series resistance ($R_G + R_S$) is equal to $R_{in}$. We can measure $V_{in}$ with a meter, but the meter resistance can "load" the circuit unless the meter resistance is much higher than the high $R_{in}$. But we can also measure $V_{out}$, because when $V_{in}$ is cut in half, $V_{out}$ is also cut in half.

To measure $R_{in}$ monitor $V_{out}$ on the scope and on the A.C. meter. Adjust the 1M POT until $V_{out}$ is cut in half. Estimate the resistance between points A and B.

| $R_{in}$ = |
|---|

7. Power Gain is now determined. With an input voltage of $V_{in}$ applied to an amplifier with $R_{in}$, the input Power is:

$$P_{in} = \frac{V^2}{R} = \frac{V_{in}^2}{R_{in}}$$

The maximum output power is obtained when the load resistor, $R_L$, equals $R_{out}$. $V_{out}$ is then cut in half. The output power is then:

$$P_{out} = \frac{V^2}{R} = \frac{\left(\frac{V_{out}}{2}\right)^2}{R_{out}}$$

where $V_{out} = K V_{in}$ (K = gain)

$$P_{out} = \frac{\left(\frac{K V_{in}}{2}\right)^2}{R_{out}} = \frac{K^2 V_{in}^2}{4 R_{out}}$$

$$\frac{P_{out}}{P_{in}} = \frac{K^2 V_{in}^2}{4 R_{out}} \times \frac{R_{in}}{V_{in}^2}$$

$$\text{POWER GAIN} = \frac{K^2}{4} \times \frac{R_{in}}{R_{out}}$$

For the Cathode Follower, use the measured gain (K), $R_{in}$ and $R_{out}$ and calculate the power gain:

CATHODE FOLLOWER POWER GAIN =

| Ed-Lab | DIODE RECTIFIER | EXP. NO. 7<br>Page No. 1 |
|---|---|---|

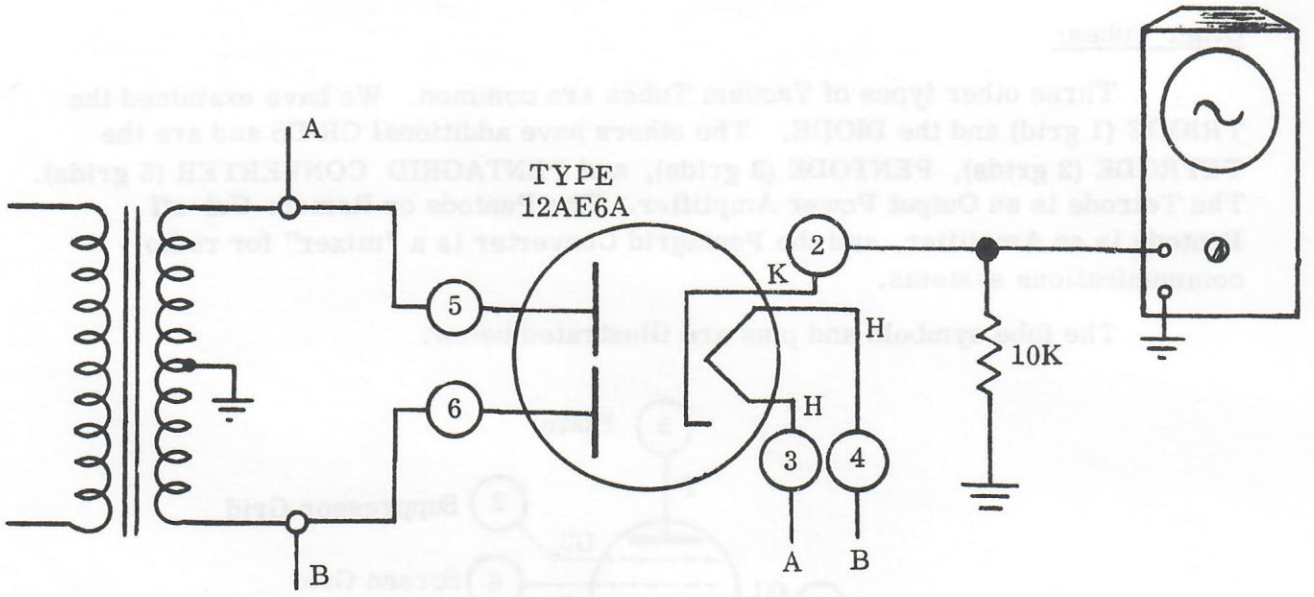


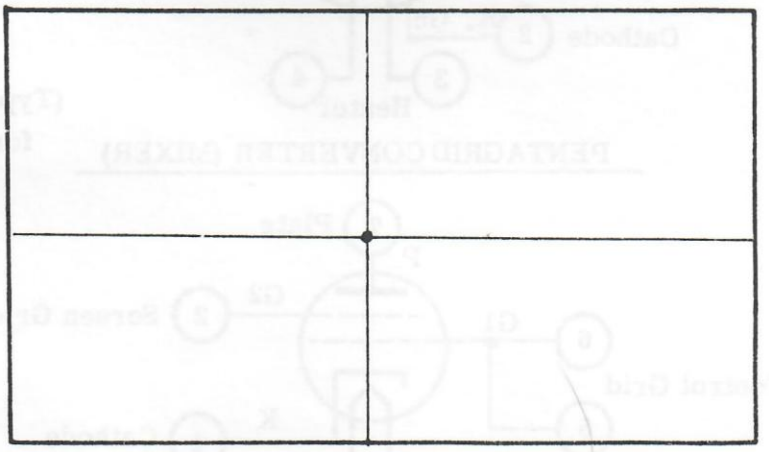

Figure 7 - 1 Vacuum Tube Full Wave Rectifier

Objective:

You will use the Vacuum Tube Diodes in a Rectifier Circuit.

Procedure:

1. Wire the circuit shown in Figure 7 - 1.
2. Monitor the cathode (K) output on the oscilloscope and sketch the full wave rectifier output.

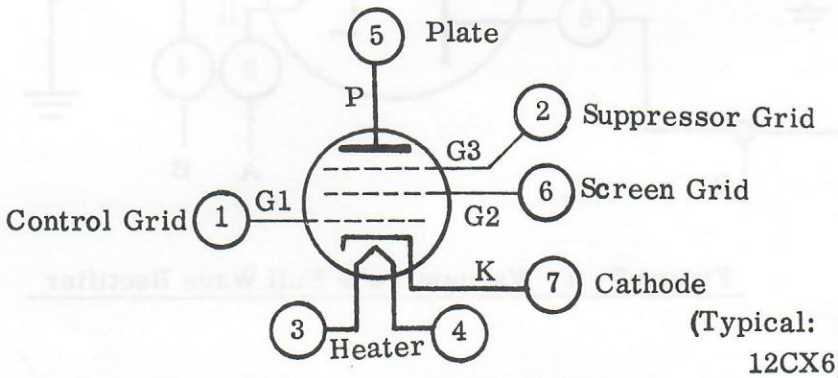


3. How does this waveform compare with the semiconductor diode full wave rectifier?

Other Tubes:

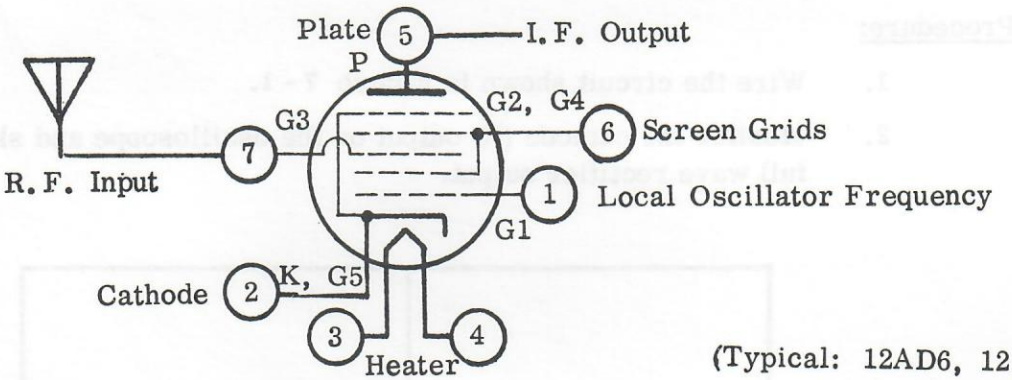
Three other types of Vacuum Tubes are common. We have examined the TRIODE (1 grid) and the DIODE. The others have additional GRIDS and are the TETRODE (2 grids), PENTODE (3 grids), and PENTAGRID CONVERTER (5 grids). The Tetrode is an Output Power Amplifier. The Pentode or Remote-Cut-off Pentode is an Amplifier, and the Pentagrid Converter is a "mixer" for radio communications systems.

The tube symbols and pins are illustrated below:



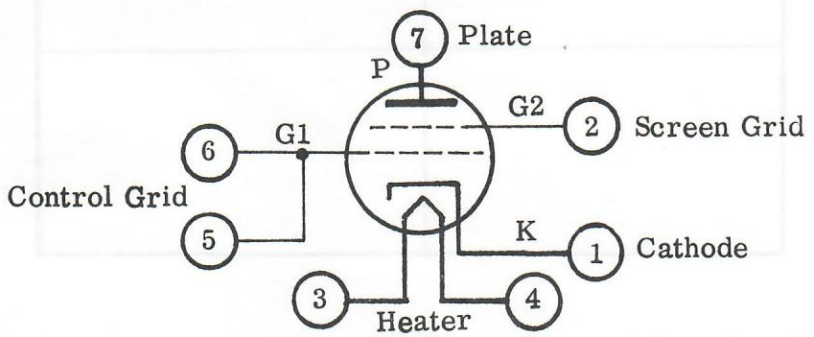

REMOTE CUT-OFF PENTODE (AMPLIFIER)

(Typical: 12BL6, 12AC6, 12CX6) are suitable for 12 volt plate voltage.




PENTAGRID CONVERTER (MIXER)

(Typical: 12AD6, 12 EG6) for 12 volt plate voltage




TETRODE

(Typical: 12K5)

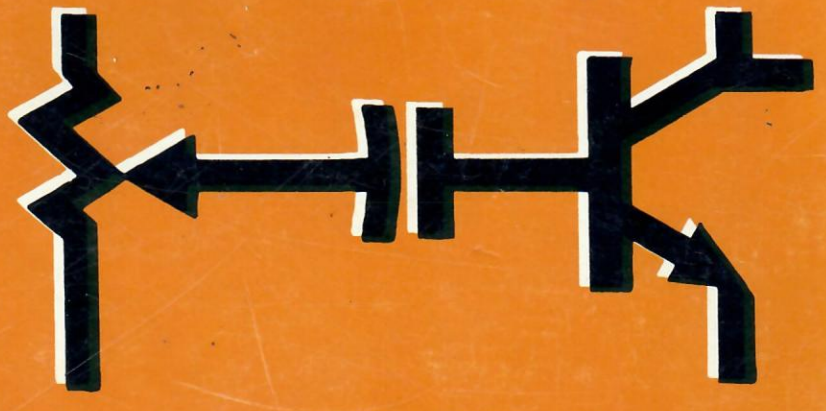
| Ed-Lab | DIODE RECTIFIER | EXP. NO. 7<br>Page No. 3 |
|---|---|---|

SUMMARY OF CHARACTERISTICS

| TYPE | | TYPE NO. | A.C. $r_p$ OHMS | $g_m$ $\mu$mhos | $I_p$ ma |
|---|---|---|---|---|---|
| TRIODES | * | 12AE6A | 13,000 | 1300 ($\mu$ = 16.7) | 1. |
| | | 12EL6 | 45,000 | 1200 ($\mu$ = 55) | 0.7 |
| TETRODE | * | 12K5 | 480 | 15,000 ($\mu$ = 7.2) | 40. |
| REMOTE CUT-OFF PENTODE | * | 12CX6 | 40K | 3,100 | 3. |
| | | 12BL6 | 500K | 1,350 | 1.35 |
| | | 12AC6 | 500K | 730 | .55 |
| PENTAGRID CONVERTER | * | 12AD6 | 1M | 260 | .45 |
| | | 12EG6 | 150K | 800 | .55 |

* Preferred Types

Ed-Lab



CES INDUSTRIES, INC.